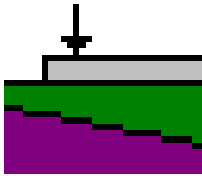


Analysis of elastically supported slabs using
subgrade reaction modulus and constrained modulus
methods and finite elements

GGU-SLAB

VERSION 9



Last revision:	October 2015
Copyright:	Prof. Dr. Johann Buß
Technical implementation and sales:	Civilserve GmbH, Steinfeld

Contents:

1	Preface	8
2	Licence protection and installation	9
3	Language selection.....	9
4	Starting the program	10
5	Short introduction	11
5.1	General introduction to working with the program.....	11
5.2	Procedure for the subgrade reaction modulus method	11
5.3	Procedure for the constrained modulus method	13
5.4	Notes applicable to both methods	15
6	Theoretical principles	16
6.1	General analysis methods.....	16
6.2	Subgrade reaction modulus method	17
6.3	Constrained modulus method.....	18
6.4	Signs and designations	19
7	Description of menu items.....	20
7.1	File menu.....	20
7.1.1	"New" menu item.....	20
7.1.2	"Load" menu item	20
7.1.3	"Save" menu item	21
7.1.4	"Save as" menu item	21
7.1.5	"Import ASCII file" menu item.....	21
7.1.6	"Export as ASCII file" menu item	22
7.1.7	"Print output table" menu item.....	22
7.1.7.1	Selecting the output format	22
7.1.7.2	Button "Output as graphics"	23
7.1.7.3	Button "Output as ASCII"	25
7.1.8	"Printer preferences" menu item.....	26
7.1.9	"Print and export" menu item	26
7.1.10	"Batch print" menu item	28
7.1.11	"Exit" menu item.....	28
7.1.12	"1, 2, 3, 4" menu items.....	28
7.2	FEM mesh menu	29
7.2.1	"Preferences" menu item.....	29
7.2.2	"FEM mesh" menu item.....	29
7.2.3	"Outline" menu item	29
7.2.4	"Define (FEM) nodes" menu item	30
7.2.5	"Change (FEM nodes)" menu item.....	31
7.2.6	"Move (FEM nodes)" menu item.....	32
7.2.7	"Edit (FEM nodes)" menu item	32
7.2.8	"Array" menu item.....	32
7.2.8.1	Select type of array.....	32
7.2.8.2	Button "Regular"	33
7.2.8.3	Button "Irregular".....	34

7.2.9	"Circular slab, ..." menu item	34
7.2.10	"Manual (FEM) mesh" menu item	36
7.2.11	"Automatic (FEM mesh)" menu item	36
7.2.12	"Round off (FEM mesh)" menu item	37
7.2.13	"Delete (FEM mesh)" menu item	37
7.2.14	"Optimize (FEM mesh)" menu item	37
7.2.14.1	"Diagonals" button	37
7.2.14.2	"Topology" button	38
7.2.15	"Align (FEM mesh)" menu item	38
7.2.16	"Refine individually" menu item	38
7.2.17	"Section" menu item	38
7.2.18	"All" menu item	39
7.2.19	Refinement methods	39
7.3	Subgrade menu (subgrade reaction modulus only)	41
7.3.1	General note on subgrade reaction moduli	41
7.3.2	"Default subgrade" menu item	41
7.3.3	"Individual subgrade" menu item	41
7.3.4	"Modify (subgr. reac. mod. at FEM nodes)" menu item	41
7.3.5	"In section" menu item	42
7.3.6	"Subgrade contours" menu item	42
7.3.7	"Interpolation mesh" menu item	42
7.3.8	"Nodes" menu item	43
7.3.9	"Mesh" menu item	43
7.3.10	"Modify" menu item	43
7.3.11	"Contours" menu item	44
7.3.12	"Import/export" menu item	44
7.3.13	"Assign" menu item	44
7.4	Constrained mod. menu (constrained modulus method only)	45
7.4.1	General note on constrained moduli	45
7.4.2	"Preferences" menu item	45
7.4.3	"Layers" menu item	46
7.4.4	"Default depths" menu item	47
7.4.5	"Layer base contours" menu item	48
7.4.6	"Edit layer depths" menu item	49
7.4.7	"Determine" menu item	49
7.4.8	"Modify" menu item	50
7.4.9	"Display in section" menu item	50
7.4.10	"Automesh" menu item	51
7.4.11	"Define nodes" menu item	51
7.4.12	"Change" menu item	52
7.4.13	"Move" menu item	52
7.4.14	"Edit" menu item	52
7.4.15	"Manual mesh" menu item	53
7.4.16	"Automatic" menu item	53
7.4.17	"Delete" menu item	53

7.4.18 "Refine" menu item.....	53
7.4.19 "Save/load" menu item	53
7.5 Boundary menu	54
7.5.1 "Preferences" menu item.....	54
7.5.2 "Check" menu item	54
7.5.3 "Single displacement + load" menu item.....	55
7.5.4 "(Displacement) In section" menu item	56
7.5.5 "Line loads" menu item	56
7.5.6 "Distributed loads" menu item.....	57
7.5.7 "Distributed loads for materials" menu item.....	57
7.5.8 "Individual materials" menu item	58
7.5.9 "(Materials) In section" menu item.....	58
7.5.10 "Beams" menu item	58
7.5.11 "Delete all" menu item.....	58
7.5.12 "Single springs" menu item	59
7.6 System menu	60
7.6.1 "Info" menu item	60
7.6.2 "Units + project identification" menu item	60
7.6.3 "Slab material" menu item	60
7.6.4 "Beam material" menu item.....	61
7.6.5 "Test" menu item	61
7.6.6 "Analyse" menu item	62
7.6.6.1 Bandwidth optimisation/equation solver selection.....	62
7.6.6.2 Analysis using the subgrade reaction modulus method.....	62
7.6.6.3 Analysis using the constrained modulus method	63
7.6.7 "Reinforce slab" menu item.....	65
7.6.8 "Footing bases" menu item	66
7.7 Graphics preferences menu	67
7.7.1 "Refresh and zoom" menu item	67
7.7.2 "Zoom info" menu item	67
7.7.3 "Pen colour and width" menu item	68
7.7.4 "Legend font selection" menu item.....	68
7.7.5 "Mini-CAD toolbar" and "Header toolbar" menu items	68
7.7.6 "Toolbar preferences" menu item	69
7.7.7 "3D toolbar" menu item.....	70
7.7.8 "General legend" menu item.....	70
7.7.9 "Materials legend" menu item.....	71
7.7.10 "Constrained mod. legend" menu item	72
7.7.11 "Reinforcement legend" menu item.....	73
7.7.12 "Beam legend" menu item	73
7.7.13 "Section legend" menu item.....	73
7.7.14 "Move legends" menu item.....	73
7.7.15 "Save graphics preferences" menu item.....	74
7.7.16 "Load graphics preferences" menu item	74

7.8	Page size + margins menu	75
7.8.1	"Auto-resize" menu item.....	75
7.8.2	"Manual resize (mouse)" menu item.....	75
7.8.3	"Manual resize (editor)" menu item.....	75
7.8.4	"Page size and margins" menu item.....	76
7.8.5	"Font size selection" menu item.....	76
7.8.6	"Margins + borders" menu item.....	77
7.8.7	"Draw Mini-CAD first" menu item	77
7.8.8	"Undo" menu item	77
7.8.9	"Restore" menu item	77
7.8.10	"Preferences" menu item.....	77
7.9	Evaluation menu.....	78
7.9.1	General information on representation of contour lines, circles and tables	78
7.9.2	"Normal contours" menu item	79
7.9.3	"Coloured" menu item	80
7.9.4	"3D" menu item	80
7.9.5	"3D array" menu item	82
7.9.6	"Circles" menu item.....	83
7.9.7	"Table" menu item.....	84
7.9.8	"Values in node section" menu item	85
7.9.9	"Position of node section" menu item.....	85
7.9.10	"Any section" menu item	85
7.9.11	"Position of any section" menu item.....	87
7.9.12	"Support" menu item.....	88
7.9.13	"Principal moments" menu item	89
7.9.14	"Reinforcement" menu item.....	90
7.9.15	"Beams" menu item	91
7.9.16	"Single springs" menu item	92
7.9.17	"Sum V" menu item.....	93
7.9.18	"Individual values" menu item.....	93
7.9.19	"Subgrade reactions" menu item.....	94
7.9.20	"Difference" menu item	94
7.9.21	"Limiting depths" menu item.....	94
7.10	Special menu (constrained modulus method only)	95
7.10.1	"Define stress section" menu item	95
7.10.2	"Preferences" menu item.....	96
7.10.3	"Stress bulb section" menu item	96
7.10.4	"(Stress bulb section) Normal contours" menu item.....	96
7.10.5	"(Stress bulb section) Coloured" menu item.....	96
7.10.6	"Vertical settlements section" menu item	97
7.10.7	"(Vertical settlements section) Normal contours" menu item.....	97
7.10.8	"(Vertical settlements section) Coloured" menu item.....	97
7.10.9	"Settlement depression" menu item	97
7.10.10	"Stresses for GGU-CONSOLIDATE" menu item.....	97

7.11 ? menu	98
7.11.1 "Copyright" menu item	98
7.11.2 "Maxima" menu item	98
7.11.3 "Help" menu item	98
7.11.4 "Test slab reinforcement" menu item	98
7.11.5 "Test vibrodisplacement compaction" menu item	99
7.11.6 "Rotate all" menu item	100
7.11.7 "GGU on the web" menu item	100
7.11.8 "GGU support" menu item	100
7.11.9 "What's new?" menu item	100
7.11.10 "Language preferences" menu item	100
7.11.11 "Benchmark" menu item (constrained modulus method only)	100
8 Tips and tricks.....	101
8.1 Keyboard and mouse	101
8.2 Function keys	102
8.3 "Copy/print area" icon	103
9 Worked slab analysis examples	104
9.1 Example 1: Analysis using the subgrade reaction modulus method	104
9.1.1 System description (Example 1)	104
9.1.2 Step 1: Select the analysis method (Example 1)	105
9.1.3 Step 2: Define the FEM mesh (Example 1)	105
9.1.4 Step 3: Optimise the FEM mesh diagonals (Example 1)	106
9.1.5 Step 4: Assign the subgrade reaction modulus (Example 1)	106
9.1.6 Step 5: Assign displacement boundary conditions (Example 1)	107
9.1.7 Step 6: Assign slab material (Example 1)	108
9.1.8 Step 7: Analyse slab (Example 1)	108
9.1.9 Step 8: Evaluation via individual values (Example 1)	109
9.1.10 Step 9: Drawing the deflection curve and other evaluations (Example 1)	110
9.2 Example 2: Slab with constant subgrade	111
9.2.1 System description (Example 2)	111
9.2.2 Step 1: Change the subgrade reaction modulus (Example 2)	111
9.2.3 Step 2: Change the boundary conditions (Example 2)	112
9.3 Example 3: Analysis using the constrained modulus profile method	113
9.3.1 System description (Example 3)	113
9.3.2 Step 1: Select the analysis method (Example 3)	113
9.3.3 Step 2: Define the constrained modulus profile (Example 3)	114
9.3.4 Step 3: Define the constrained modulus profile mesh (Example 3)	115
9.3.5 Step 4: Analyse slab (Example 3)	116
10 Index.....	117

List of Figures and Tables:

<i>Figure 1</i> Optimisation of diagonals	37
<i>Figure 2</i> FEM refinement demonstration mesh	39
<i>Figure 3</i> FEM mesh refinement using Method 1.....	40
<i>Figure 4</i> FEM mesh refinement using Method 2.....	40
<i>Figure 5</i> FEM mesh refinement using Method 3.....	40
<i>Figure 6</i> Example from Schwarz 1991.....	104
 <i>Table 1</i> Displacements at the slab centre [m]	 109
<i>Table 2</i> Displacements at the lower slab face [m].....	110

1 Preface

The **GGU-SLAB** program can analyse slabs using the subgrade reaction modulus and the constrained modulus methods. The finite-element method using triangle elements is employed to solve the differential equation. The subgrade modulus profile can be varied as required with almost no constraints. The constrained modulus distribution can be defined via stratigraphic profiles and an interpolation mesh. This allows any imaginable stratigraphy to be generated. The influence of vibrodisplacement compaction after Priebe can be investigated.

Further important capabilities include:

- limiting depth analysis to DIN 4019 for constrained modulus method,
- any type of slab boundary including with holes,
- linearly variable subgrade within an element,
- beam elements for modelling footing beams, for example,
- single springs, perpendicular to slab, e.g. for modelling piles.

The program system includes a powerful mesh generator and easy-to-use routines for comfortable evaluation of the analysis results (contours, 3D graphics, etc.).

The application is designed to allow simple data input or modification. The input is immediately shown on the screen, giving you optimum control over what you are doing. Graphics output supports the true-type fonts supplied with WINDOWS, so that excellent layout is guaranteed. Colour output and any graphics (e.g. files in formats BMP, JPG, PSP, TIF, etc.) are supported. DXF files can also be imported by means of the integrated Mini-CAD module (see the "**Mini-CAD**" manual).

It is not the aim of this manual to offer an introduction to the finite-element method. For details of the finite-element method, please see O. C. Zienkiewicz, "*Methode der Finiten Elemente*" ("Finite-Element Methods"), Carl Hanser Verlag Munich, Vienna, 1984.

The program system has been used in a large number of projects and has been thoroughly tested (using analytical solutions and in comparison with other FEM applications). No faults have been found. Nevertheless, liability for completeness and correctness of the program and the manual, and for any damage resulting from incompleteness or incorrectness, cannot be accepted.

2 Licence protection and installation

In order to guarantee a high degree of quality, a hardware-based copy protection system is used for the **GGU-SLAB** program.

The GGU software protected by the *CodeMeter* copy protection system is only available in conjunction with the *CodeMeter stick* copy protection component (hardware for connection to the PC, "*CM stick*"). Because of the way the system is configured, the protected software can only be operated with the corresponding CM stick. This creates a fixed link between the software licence and the CM stick copy protection hardware; the licence as such is thus represented by the CM stick. The correct Runtime Kit for the CodeMeter stick must be installed on your PC.

Upon start-up and during running, the **GGU-SLAB** program checks that a CM stick is connected. If it has been removed, the program can no longer be executed.

For installation of GGU software and the CodeMeter software please refer to the information in the *Installation notes for GGU Software International*, which are supplied with the program.

3 Language selection

GGU-SLAB is a multilingual program. The program always starts with the language setting applicable when it was last ended.

The language preferences can be changed at any time in the "?" menu, using the menu item "**Language preferences**" (in German: "**Spracheinstellung**", in Spanish: "**Configuración de idioma**").

4 Starting the program

After starting the program, you will see two menus at the top of the window:

- File
- ?

By going to the **"File"** menu, a previously analysed system can be loaded by means of the **"Load"** menu item, or a new one created using **"New"**. The program allows simple input procedures by moving directly to a dialog box after **"New"** is clicked; this can also be reached by pointing to the **"File/new"** menu item. You can now directly select the analysis method and the standard to be adopted (see Section 7.1.1). After confirming your inputs, you then see nine or ten menus in the menu bar:

- File
- FEM mesh
- Subgrade/Constrained mod.
- Boundary
- System
- Graphics preferences
- Page size + margins
- Evaluation
- Special (only with Constrained modulus method)
- ?

After clicking one of these menus, the so-called menu items roll down, allowing you access to all program functions.

The program works on the principle of ***What you see is what you get***. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the **GGU-SLAB** screen is not refreshed after every alteration.

If you would like to refresh the screen contents, press either **[F2]** or **[Esc]**. The **[Esc]** key additionally sets the screen presentation back to your current zoom, which has the default value 1.0, corresponding to an A3 format sheet.

5 Short introduction

5.1 General introduction to working with the program

Since I know from personal experience that reading user-manuals is quite a chore, there will now follow a short description of the main program functions. After studying this section you will be in a position to carry out analysis of a slab using the finite-element method. Detailed information on the program can be obtained from the subsequent chapters as you require it.

Because the program can perform an analysis using both the subgrade reaction modulus and the constrained modulus methods, the descriptions are kept separate. Even if you want to stick to the constrained modulus method only, you should still read the following section on the use of the subgrade reaction modulus method. In order to get some practice we recommend analysing a slab using the subgrade reaction modulus method. The input for this method is somewhat simpler, meaning you can get to know the program without a lot of unnecessary ballast.

Beside this short description, the final section of this manual contains a practical example, explaining in detail the use of the program.

5.2 Procedure for the subgrade reaction modulus method

- Design the system to be analysed.
- Start the **GGU-SLAB** program and go to the **"File/New"** menu item. Select the type of system (here: subgrade reaction modulus method) and the safety concept.
- If necessary, adapt the page coordinates to your system. To do this, go to the **"Page size + margins/Manual resize (editor)"** menu item.
- Then select the **"FEM mesh/Define nodes"** menu item.
- Click the governing nodes (points) in your system using the mouse. The points are numbered. Alternatively, you can enter the system nodes in a table using the menu item **"FEM mesh/Edit"**. When defining new nodes the new node is assigned the *default subgrade reaction modulus*. This can be edited in the **"Subgrade/Default subgrade"** menu item.
- If the nodes are outside of the page coordinates, go to the menu item **"Page size + margins/Auto-resize"** or use the [F9] function key.
- Then go to the **"FEM mesh/Manual mesh"** menu item and join groups of three nodes to form triangle elements. You create the coarse structure for your system in this manner. Alternatively, you can utilise the **"FEM mesh/Automatic"** menu item to have the program do this job for you.
- If you want to alter the locations of mesh nodes, go to the menu item **"FEM mesh/Edit"**, **"FEM mesh/Change"** or **"FEM mesh/Move"**.
- If you need to analyse a simple rectangular slab with a varying model point array, you can use the menu item **"FEM mesh/Array"** ("irregular" button) to perform the mesh generation in just a few seconds.
- If you need to alter the subgrade reaction moduli, use the menu items **"Subgrade/Individual subgrade"**, **"Subgrade/Modify"** or **"Subgrade/In section"**. Using **"Subgrade/Subgrade contours"** it is possible to view the selected subgrade reaction moduli distributions.
- If you need to delete a triangle element, select the menu item **"FEM mesh/Manual mesh"** once again and click the three corner nodes of the appropriate element. In this menu item, try double-clicking in a triangle element.

- If you need to assign different material properties to triangle elements (e.g. a different slab thickness), use the menu item **"Boundary/Individual materials"** or **"Boundary/(Materials) In section"** and assign different material numbers for one or more elements. An input line appears for each material number in the **"System/Slab material"** menu item.
- The screen contents can be refreshed at any time using the [ESC] or [F2] keys.
- You can create a finer structure for your system from the coarse structure, by selecting the menu items **"FEM mesh/Refine individually"**, **"FEM mesh/Section"** or **"FEM mesh/All"**.
- You can edit the system as required even after refining your mesh by using **"FEM mesh/Define nodes"**, **"FEM mesh/Manual mesh"**, etc.
- Whatever you do, in order to acquire a numerically favourable FEM mesh the menu item **"FEM mesh/Optimise"**, and here the **"Diagonals"** button in particular, should be used.
- You can create acute angled, and thus numerically unfavourable, triangle elements for demonstration purposes using the menu item **"FEM mesh/Move"**. Then select the **"FEM mesh/Optimise"** menu item (**"Topology"** button) and follow the effects on the screen.
- Define the governing boundary conditions for your system, e.g. via the menu items **"Boundary/Single displacement"** or **"Boundary/(Displacement) In section"**.
- Define the governing action boundary conditions for your system, e.g. via the menu items **"Boundary/Point loads"** or **"Boundary/Line loads"**.
- If needed, edit the material numbers using the **"Boundary/Individual materials"** menu item.
- If needed, edit the material properties using the **"System/Slab material"** menu item.
- If beam elements are used in the system (e.g. joists), define the location of beam elements using the menu item **"Boundary/Beams"** by placing a *section* along the required node. After pressing the [Return] key you can assign the defined beams a material number. The beam stiffnesses (EJ and GJ) can then be edited in **"System/Beam materials"**.
- If you need to simulate the influence of single springs perpendicular to the slab (e.g. piles or slightly yielding supports), go to the menu item **"Boundary/Single springs"** and click the system nodes to be assigned single springs. Then enter the spring constant.
- A simple check of the defined boundary conditions is possible by going to the **"Boundary/Check"** menu item.
- Once mesh generation is complete, go to the **"System/Analyse"** menu item and start the analysis. If necessary, the program performs a bandwidth optimisation before carrying on with the analysis, in order to acquire a numerically favourably configured equation system. After bandwidth optimisation the program asks which equation solver you want to use. If you are not sure which of the two to use, click **"Equation solver info"**. If you want to rule out a disallowed tension bedding for certain soils, activate the appropriate check box in the *start dialog box*. Once the analysis is complete the program then checks whether bedded slab areas are lifted. The subgrade reaction modulus is set to **"0"** in these areas and the analysis performed once again. This process is carried on iteratively until no more bedded slab areas are subject to tension.
- Once analysis is complete you can output the results as a log to the printer or save them to a file, as preferred. In general though, this type of result presentation is less than satisfactory.

- Instead, go immediately to the **"Evaluation"** menu. A variety of evaluation options are provided here. The menu items **"Evaluation/Coloured"** or **"Evaluation/3D"** are particularly impressive. The dialog boxes which then appear can almost always be closed using **"OK"**, without further changes having to be made. The program will generally suggest sensible input data. The **"Determine extreme values"** button should be clicked once, however, otherwise an error message containing a correction suggestion will appear.
- If you have a WINDOWS colour printer installed you can create colour output by going to **"File/Print and export"** and then pressing the **"Printer"** button in the subsequent dialog box. Grey scale is used for mono printers.
- Experiment a little with the examples provided.

5.3 Procedure for the constrained modulus method

Definition of the FEM mesh follows the same procedure described for the subgrade reaction modulus method. Please read Section 5.2 first. The **"Subgrade"** menu is now replaced by the **"Constrained mod."** menu.

It is necessary to define a triangle mesh for an analysis using the constrained modulus method. This mesh is not related to the FEM mesh. The nodes of this triangle mesh are formed by drilling points. The drilling points are connected to form a triangle mesh, allowing the program to interpolate the constrained modulus profile at each point. Proceed as follows to define the drilling points:

- First select the menu item **"Constrained mod./Layers"**. You can edit the soil properties and the number of layers in the dialog box. Click the **"Edit number of layers"** button and enter the new number of layers.
- You should then (not absolutely necessary) go to the menu item **"Constrained mod./Default depths"**. If you have defined three layers in **"Constrained mod./Layers"**, this dialog box displays three layer depths. The associated soil properties are also listed for your information. The depths given can be edited to suit your requirements. The depths are given as depths in metres below grade (m bg). When subsequently defining triangle nodes these layer depths are assigned to the corresponding node. You can then edit the layer depths as required for each node. However, if the constrained modulus profile is the same for almost all nodes of the triangle mesh, you can save some input effort by initially defining default depths. If you click the **"For all"** button, all existing triangle nodes are assigned these depths.
- After making these preparations you can specify the locations of triangle nodes (drilling points). A coordinate system is visible on the screen. If the field of view does not correspond to your basic system go to the menu item **"Page size + margins/Manual resize (editor)"** and enter the values for your system in the dialog box.
- Then go to the menu item **"Constrained mod./Define nodes"** (not **"FEM mesh/Define nodes"**!). Using the left mouse button, click the triangle nodes (= location of points at which the constrained modulus profile is known). The current coordinates of the tip of the mouse pointer are shown in the program window title bar. Input entered by mistake can be undone by right-clicking the node. If you press the [ESC] or the [F2] key, the screen is refreshed and you will see a graphical representation of the constrained modulus profiles.
- If this representation appears too small or too large, go to the **"Constrained mod./Preferences"** menu item and adjust the factors for representation of the constrained modulus profile width and depth to suit your needs.
- Alternatively to coordinate input using the mouse, you can enter the values in a table. To do this, select the menu item **"Constrained mod./Edit"**.

- After node input you can edit the depths of the of the individual layers at the nodes. To do this, select the menu item "**Constrained mod./Edit layer depths**". Once this is done, a double-click near a node is sufficient to open the following dialog box:

No	Base [m bg]	Es [MN/m²]	gamma [kN/m³]	nu [-]	Designation
1	3.00	1.00	14.00	0.000	Clay
2	7.00	20.00	9.00	0.000	Sand, silty

This box is absolutely identical to the box for the default depths. The only difference is that any alterations are with regard to the current node. The depths are given as depths in metres below grade.

- Once you have defined at least 3 nodes (drilling points) you must connect these nodes to a triangle mesh in order to allow the program to carry out interpolation during the analysis. You have two options:

"Constrained mod./Manual mesh"

Click three nodes to create each respective triangle. Mistakenly entered triangles can be deleted again by clicking the three nodes.

"Constrained mod./Automatic"

The program carries out a so-called triangulation and connects all triangles to form a triangle mesh. Even after this you still have the option of deleting triangles by clicking the three nodes of a triangle using "**Constrained mod./Manual mesh**". If a triangle mesh already exists you will be asked before triangulation whether you want to delete this mesh. Only press the "**Supplement**" button in exceptional cases. The triangulation follows certain laws which may not allow sensible enhancement of an existing partial mesh.

The constrained modulus method requires an iteration process, in contrast to the subgrade reaction modulus method. In the first step, the settlements at all FEM mesh nodes resulting from a constant load of 1 kN/m² on the FEM elements are calculated. To do this, the Boussinesq equation is numerically integrated, because there is currently no analytical solution to this problem. The pressures exercised on the nodes (in the first step = 1 kN/m²; in all following iteration steps = $k_s \cdot w$) are divided by the calculated settlements in order to acquire the subgrade reaction modulus for each node. Following this a calculation is performed using the subgrade reaction modulus method including determination of the node displacements. If the slab differential settlements deviate from the user-defined settlement given at the start of analysis, the iteration is continued.

The constrained modulus method employs a limiting depth corresponding to the base of the constrained modulus profile. Alternatively, the limiting depth can be calculated to DIN 4019. To do this, the appropriate check box must be activated in the "**System/Analyse**" dialog box (see Section 7.6.6.3).

5.4 Notes applicable to both methods

Once the analysis is complete the slab can be reinforced to EC 2/DIN 1045 specifications (menu item "**System/Reinforce slab**"). The support forces calculated here are adopted for the shear design at support points. They are the result of the first derivation of the moments.

Note:

Because the derivation causes a roughening of the function, shear force calculation represents a fundamental problem when using the finite-element method. Remember that all finite-element methods are approximation methods. The quality of the approximation increases with increasing mesh density. In the current version, systems with a maximum of 45,000 triangle elements and nodes can be processed.

This short description shows that the analysis of a slab only requires selection of a very few menu items. All other menu items deal principally with saving your data, the layout and, if required, further evaluation of the model. These are described in the following chapters.

6 Theoretical principles

6.1 General analysis methods

An analytical solution is only possible for simple systems. We must rely on numerical solution methods when modelling complex systems. These primarily include the:

- finite-difference (FDM) and
- finite-element (FEM)

methods. When using finite methods, the total area is subdivided into many small (finite) sub-areas (elements). FEM generally employs triangles for these small areas. Simple, generally quadratic approximation functions are used within these triangles. The real, complex, mosaic-like overall solution is composed of the numerous simple partial solutions. This gives rise to equation systems comprised of a number of unknowns corresponding to the number of system nodes. Normally, using the finite difference method, the only option for discretising the total area is by means of rectangular sub-areas. In contrast to FEM, then, FDM is considerably more flexible with regard to adaptation to complicated boundary structures. In addition, mesh refinement is not as easy to perform in some areas. The resulting equation systems are also numerically more stable for FEM. The main advantage of FDM consists only of the theoretically less complex basic mathematical relationships. These will generally be of little interest to the program user. The **GGU-SLAB** program uses the finite-element method.

When using this program, please remember that all finite element or finite difference methods are approximation methods. The quality of the approximation in terms of the actual solution increases as the mesh refinement increases. Take care to ensure that the mesh is denser in those areas where the principal forces act (e.g. support points, point loads). The shape of the triangles also exercises a certain influence. Optimum conditions are achieved with equilateral triangles. You can get an idea of the quality of the solution by modelling the same system again, but using either a finer or a coarser mesh and then comparing the results of the two models.

The following general comments on the **GGU-SLAB** program are also important:

- triangle elements are used,
- Hooke's Law applies.

6.2 Subgrade reaction modulus method

The program solves the differential equation:

$$\frac{d^4 w}{dx^4} + 2 \frac{d^4 w}{dx^2 dy^2} + \frac{d^4 w}{dy^4} + (q + k_s \cdot w) \frac{12(1 - \nu)^2}{Et^3} = 0$$

Where:

- w = deflection of the slab [m]
- x, y = coordinates [m]
- k_s = modulus of subgrade reaction [kN/m³]
- q = distributed load [kN/m²]
- ν = Poisson's ratio of slab [-]
- E = Young's modulus [kN/m²]
- t = thickness of slab [m]

Analytical solutions for this differential equation only exist for a few special cases. For problems in daily design practice (with variously distributed loads, freely pivoted, restrained or supported boundaries, etc.) numerical methods must be relied on.

The differential equation is solved by the program using finite-element-methods. Triangle elements are used. Simple assumptions are made for these triangle elements with regard to displacements. In the present case, a displacement assumption is used that is described in Zienkiewicz (Carl-Hanser-Verlag, 1984, Page 236). This displacement approach produces very good results compared to other approaches. The approach adopted leads to equation systems with a number of variables corresponding to three times the number of system nodes. The overall solution is assembled like a mosaic from the many partial solutions via the triangle elements. It is clear that the quality of the solution is increased with increasing finite-element mesh refinement.

The moment distribution is acquired from the two-fold numerical differentiation of the deflection area. The numerical differentiation always produces an undesirable roughening effect. In order to compensate for this the program offers two different methods for determination of moments:

- **Method 1:**
The moments are determined in the element centre in a post-processing calculation and then proportionally distributed across the neighbouring nodes. Generally delivers the best values. The moments may deviate, but only in the boundary regions.
- **Method 2:**
The moments are determined in the triangle nodes of each element in a post-processing calculation. The actual value at each node is given by forming the mean. Only produces better values than Method 1 (boundary region) for *simple* systems.

Shear force calculation represents a further fundamental problem when using the finite-element method because the derivation of the moments cause additional roughening of the function.

The quality of the calculated displacements is, generally, excellent. If you are only interested in the displacements you need not worry about the following explanations.

Remember that all finite-element methods are approximation methods. The quality of the approximation increases with increasing mesh density. In the current version, systems with a maximum of 45,000 triangle elements and nodes can be processed.

Note on boundary conditions:

The case of a boundary with a free-earth support is automatically considered when the finite-element-method is employed. All system boundaries or partial system boundaries possessing no action or displacement boundary conditions automatically have a free-earth support. In finite-element theory this type of boundary condition is also known as a natural boundary condition.

6.3 **Constrained modulus method**

The following condition must be met in order to employ the constrained modulus method:

Deflection curve of slab = settlement depression of elastic-isotropic half-space

The constrained modulus method therefore requires an iteration process, in contrast to the subgrade reaction modulus method. In the first step, the settlements (elastic-isotropic half-space) at all FEM mesh nodes resulting from a constant load of 1 kN/m² on the FEM elements are calculated. The settlement resulting from the triangle loads must be determined for every FEM mesh node. For example, a mesh comprising 512 triangles and 289 nodes requires

$$512 \cdot 289 = 147,968 (!) \text{ settlement calculations.}$$

The Boussinesq equation is numerically integrated for the settlement calculation because there is currently no analytical solution to this problem. The pressures exercised on the nodes (in the first step = 1 kN/m²; in all following iteration steps = $k_s \cdot w$) are divided by the calculated settlements in order to acquire the subgrade reaction modulus for each node. Following this a calculation is performed using the subgrade reaction modulus method including determination of the node displacements. If the slab differential settlements deviate from the user-defined settlement given at the start of analysis, the iteration is continued.

The constrained modulus method employs a limiting depth corresponding to the base of the constrained modulus profile. Alternatively, the limiting depth can be calculated to DIN 4019. To do this, the appropriate check box must be activated in the "**System/Analyse**" dialog box (see Section 7.6.6.3).

6.4 Signs and designations

The following rules apply to signs:

- Downward displacement w [m] positive.
- Rotation $\phi(x)$ [radians¹] = rotation around the x-axis is positive clockwise (direction of view for increasing values of x)
- Rotation $\phi(y)$ = rotation around the y-axis is positive clockwise (direction of view for increasing values of y)
- Point load P [kN] positive downwards
- Moment m_{xx} = moment generating stresses in x-direction, positive clockwise (direction of view for increasing values of y)
- Moment m_{yy} = moment generating stresses in y-direction, positive clockwise (direction of view for increasing values of x)

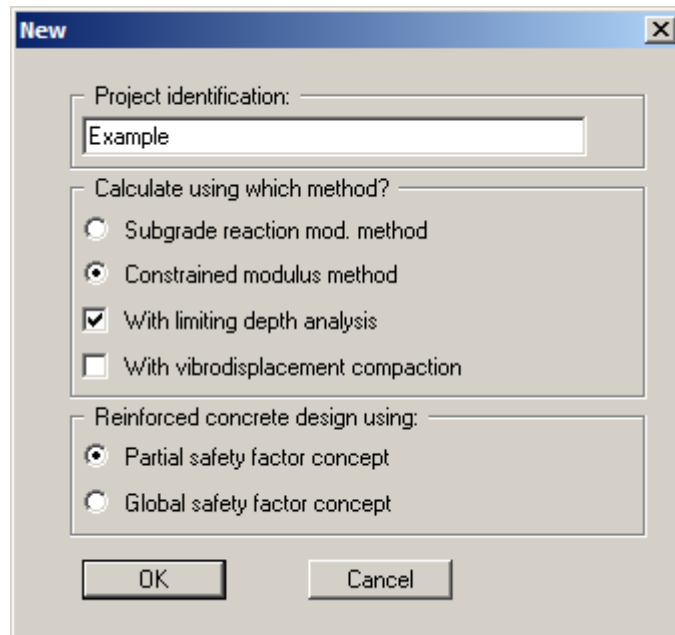
¹ Radians: For example, a value of 0.02 means $0.02 \cdot 180/\pi = 1.15^\circ$

7 Description of menu items

7.1 File menu

7.1.1 "New" menu item

You can enter a new system using this menu item. You will see the following dialog box:



You can enter a description of the system being worked on; this will then be used in the *General legend* (see Section 7.7.8).

First, specify the analysis method in the dialog box. A limiting depth analysis can be made when using the constrained modulus method. The limiting depth analysis can also be activated in the dialog box in the menu item "**System/Analyse**". Further preferences for the analysis can be specified there (see Section 7.6.6.3).

Activate vibrodisplacement compaction adoption after Priebe (Heinz J. Priebe, *Die Bemessung von Rüttelstopfverdichtung*, Ground Engineering, December 1995) using the "**With vibrodisplacement compaction**" check box.

You can select the safety concept to be employed for analysis and design of the reinforced concrete. Activate the standard required using the corresponding command button.

If an FEM mesh has already been entered and you want to switch from the subgrade reaction modulus method to the constrained modulus method (or vice versa), retaining the same mesh, you can elect to retain the existing FEM mesh after leaving the dialog box using "**OK**".

7.1.2 "Load" menu item

You can load a file with system data, which was created and saved at a previous sitting, and then edit the data.

7.1.3 "Save" menu item

You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved without prompting with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

7.1.4 "Save as" menu item

You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use ".pla" as file suffix, as this is the suffix used in the file requester box for the menu item "File/Load". If you choose not to enter an extension when saving, ".pla" will be used automatically.

If, at the time of saving, the current system has been analysed, you can decide in a dialog box whether to save the analysis results with the document. If you select "No" here you must reanalyse the file if it is opened again later.

7.1.5 "Import ASCII file" menu item

If the coordinates of the FEM mesh nodes are available in ASCII file format, they can be imported into the program. Each row of the file must contain the x- and y-value of a node. Decimal fractions must use a point, not a comma. When importing the ASCII file you must specify the columns containing the x- and the y-values.

Import ASCII

File: C:\Program Files\GGU-Software\GGU-SLAB_9\GGU-SLAB.txt

Row 2 of 868 rows

P1 -2.48977774788672E+0001 -7.76457135307566E-0001 1.15394304273056E+0001

Column delimiter:

- ☒ Space
- ☐ Tab
- ☐ Semicolon
- ☐ Comma

Data

Column x: 2 Column y: 3

Import first valid row

No. of valid rows

Result:

-2.489777747886 -7.764571353075

Import data Cancel

The current line of the ASCII file is shown at the top. You can navigate through the file using the arrow buttons on the right. If all the information is correct, the result for the row is shown in the box below the column. Otherwise, an error message appears. You may need to change the column delimiter. If the file contains invalid as well as valid rows, these will simply be skipped during the subsequent import. Finally, select the **"Import data"** button. The imported coordinates can then be processed to form an FEM mesh.

7.1.6 "Export as ASCII file" menu item

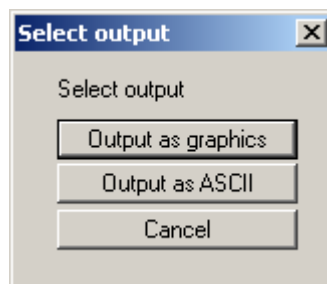
If an FEM mesh has been generated, the node coordinates can be saved to an ASCII file, allowing them to be imported into other programs where required.

7.1.7 "Print output table" menu item

7.1.7.1 *Selecting the output format*

You can have a table printed containing the current analysis results. The results can be sent to the printer or to a file (e.g. for further editing in a word processor). The output contains all information on the current state of analysis, including the system data.

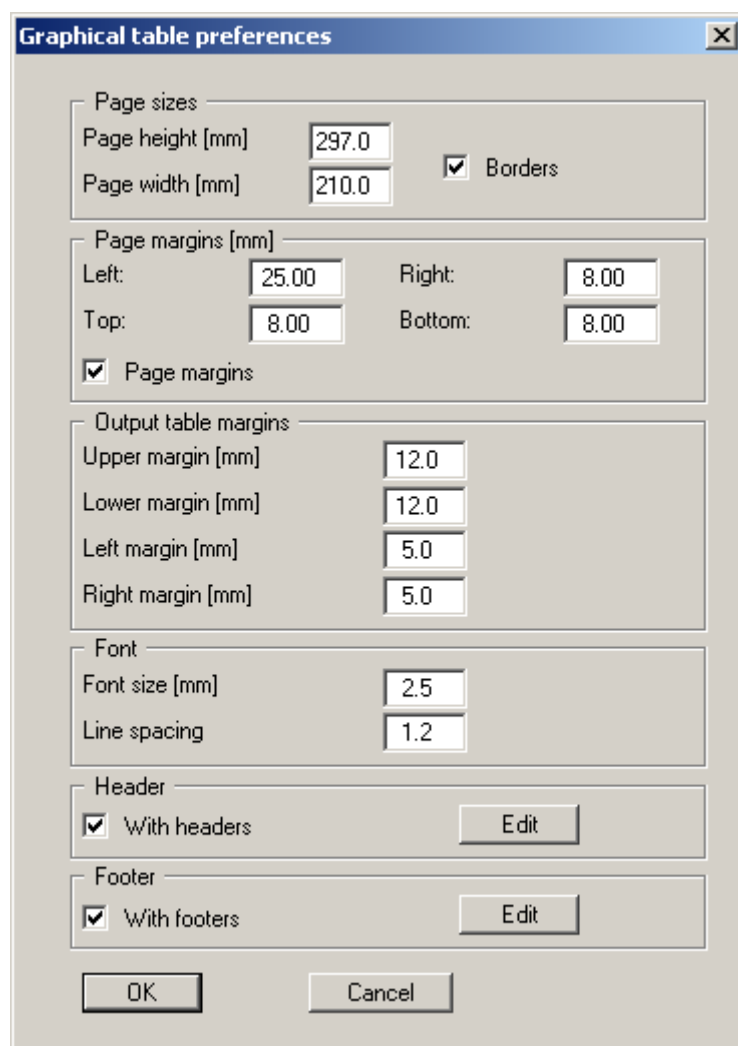
You have the option of designing and printing the output table as an annex to your report within the **GGU-SLAB** program. To do this, select **"Output as graphics"** from the following options.



If you prefer to easily print or process the data in a different application, you can send them directly to the printer or save them to a file using the **"Output as ASCII"** command button.

7.1.7.2 Button "Output as graphics"

If you selected the "Output as graphics" button in the previous dialog box a further dialog box opens, in which you can define further preferences for result visualisation.



The dialog box is titled "Graphical table preferences" and contains several sections for configuring the output table's appearance. The "Page sizes" section includes input fields for "Page height [mm]" (297.0) and "Page width [mm]" (210.0), along with a checked "Borders" checkbox. The "Page margins [mm]" section has input fields for "Left" (25.00), "Right" (8.00), "Top" (8.00), and "Bottom" (8.00), with a checked "Page margins" checkbox. The "Output table margins" section includes input fields for "Upper margin [mm]" (12.0), "Lower margin [mm]" (12.0), "Left margin [mm]" (5.0), and "Right margin [mm]" (5.0). The "Font" section has input fields for "Font size [mm]" (2.5) and "Line spacing" (1.2). The "Header" section has a checked "With headers" checkbox and an "Edit" button. The "Footer" section has a checked "With footers" checkbox and an "Edit" button. At the bottom are "OK" and "Cancel" buttons.

Page sizes	
Page height [mm]	297.0
Page width [mm]	210.0
<input checked="" type="checkbox"/> Borders	

Page margins [mm]	
Left:	25.00
Right:	8.00
Top:	8.00
Bottom:	8.00
<input checked="" type="checkbox"/> Page margins	

Output table margins	
Upper margin [mm]	12.0
Lower margin [mm]	12.0
Left margin [mm]	5.0
Right margin [mm]	5.0

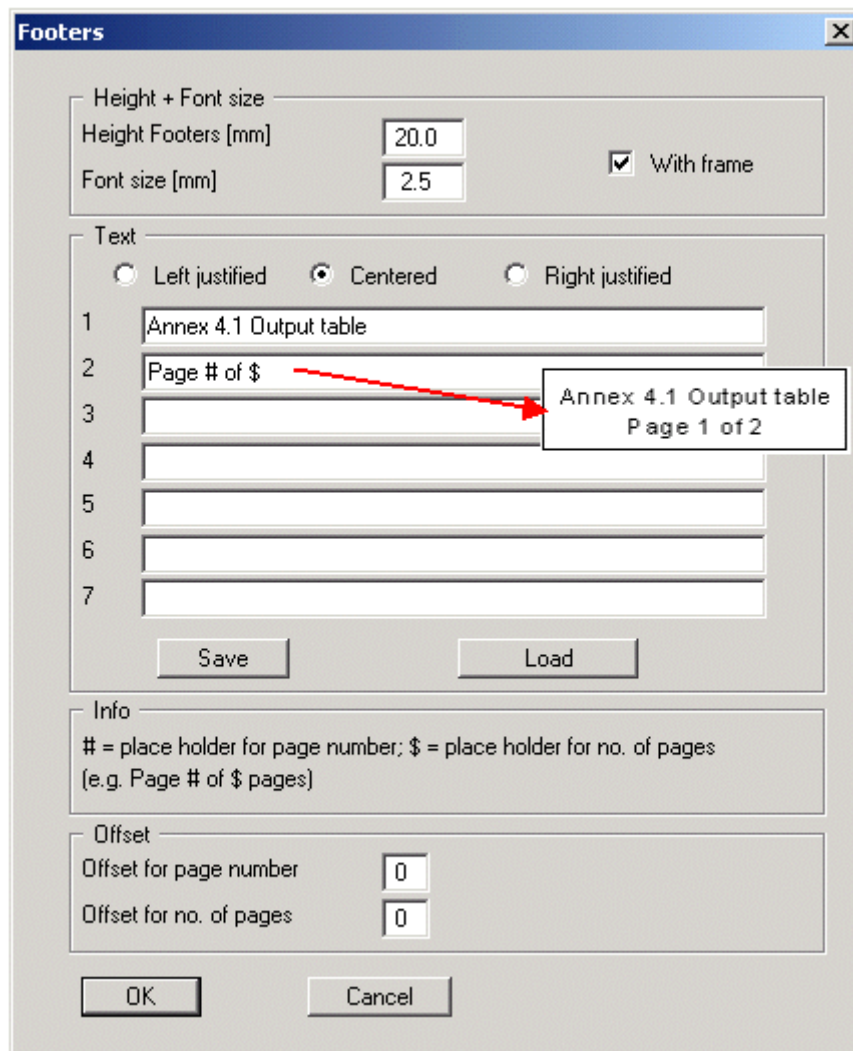
Font	
Font size [mm]	2.5
Line spacing	1.2

Header	
<input checked="" type="checkbox"/> With headers	Edit

Footer	
<input checked="" type="checkbox"/> With footers	Edit

OK Cancel

You can define the desired layout for the output table in various areas of the dialog box. If you need to add a header or footer (e.g. for page numbering), activate the appropriate check boxes "With headers" and/or "With footers" and click on the "Edit" button. You can then edit as required in a further dialog box.



Footers

Height + Font size

Height Footers [mm]

Font size [mm]

☒ With frame

Text

☐ Left justified ☒ Centered ☐ Right justified

1

2

3

4

5

6

7

Info




= place holder for page number; \$ = place holder for no. of pages
(e.g. Page # of \$ pages)

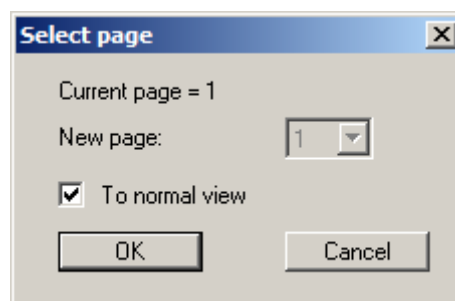
Offset

Offset for page number

Offset for no. of pages

Note: A red arrow points from the 'Page # of \$' text in line 2 to a preview box showing 'Annex 4.1 Output table' and 'Page 1 of 2'.

Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "OK" you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the "Start" button the output table is presented on the screen page by page. To navigate between the pages, use the arrow tools   in the toolbar. If you need to jump to a given page or back to the graphical representation, click on the  tool. You will then see the following box:



Select page

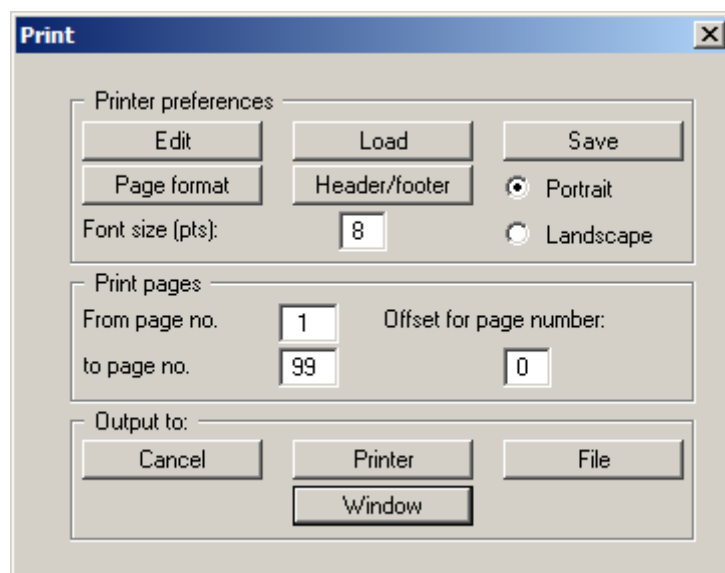
Current page = 1

New page:

☒ To normal view

7.1.7.3 Button "Output as ASCII"

You can have your calculation data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application. After selecting the button **"Output as ASCII"** you will see a further dialog box in which you can select the parameters to be used. If you click the **"Start"** button, the following dialog box appears in which you can define output preferences.



In the dialog box you can define output preferences:

- **"Printer preferences"** group box
Using the **"Edit"** button the current printer preferences can be changed or a different printer selected. Using the **"Save"** button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select **"GGU-SLAB.drk"** as file name and save the file in the program folder (default), the file will be automatically loaded the next time you start the program.

Using the **"Page format"** button you can define, amongst other things, the size of the left margin and the number of lines per page. The **"Header/footer"** button allows you to enter a header and footer text for each page. If the **"#"** symbol appears within the text, the current page number will be entered during printing (e.g. **"Page #"**). The text size is given in **"Pts"**. You can also change between **"Portrait"** and **"Landscape"** formats.
- **"Print pages"** group box
If you do not wish pagination to begin with **"1"** you can add an *offset number* to the check box. This offset will be added to the current page number. The output range is defined using **"From page no."** **"to page no."**.
- **"Output to:"** group box
Start output by clicking on **"Printer"** or **"File"**. The file name can then be selected from or entered into the box. If you select the **"Window"** button the results are sent to a separate window. Further text editing options are available in this window, as well as loading, saving and printing.

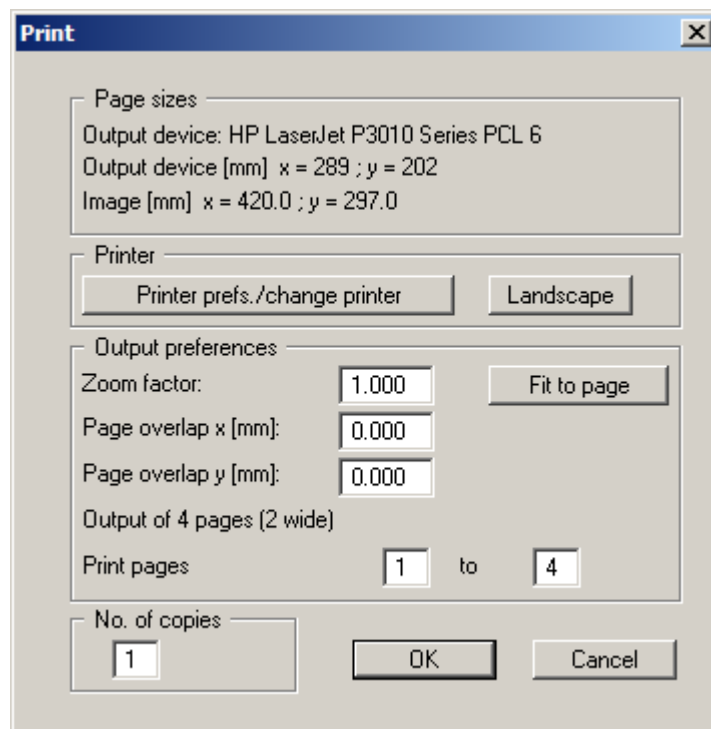
7.1.8 "Printer preferences" menu item

You can edit printer preferences (e.g. swap between portrait and landscape) or change the printer in accordance with WINDOWS conventions.

7.1.9 "Print and export" menu item

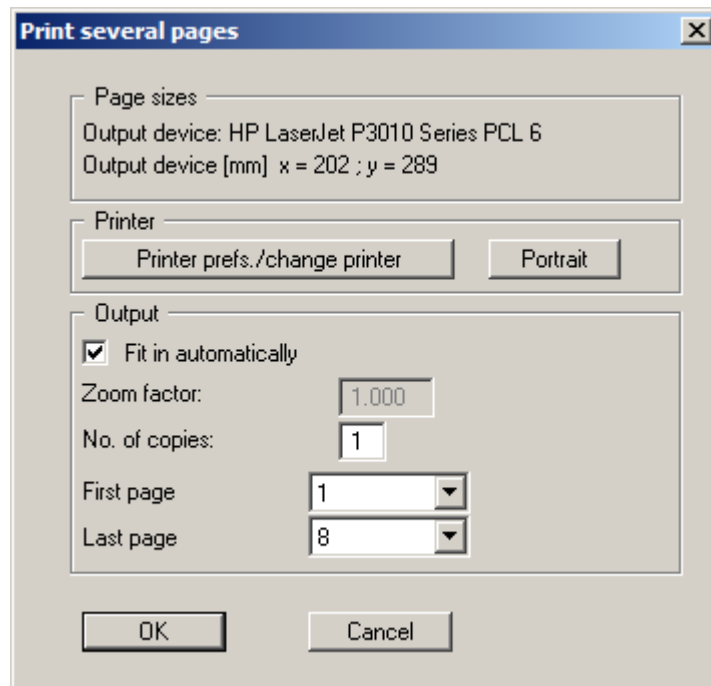
You can select your output format in a dialog box. You have the following possibilities:

- **"Printer"**
allows graphic output of the current screen contents (*graphical representation*) to the WINDOWS standard printer or to any other printer selected using the menu item **"File/Printer preferences"**. But you may also select a different printer in the following dialog box by pressing the **"Printer prefs./change printer"** button..




In the upper group box, the maximum dimensions which the printer can accept are given. Below this, the dimensions of the image to be printed are given. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 4). In order to facilitate better re-connection of the images, the possibility of entering an overlap for each page, in x and y direction, is given. Alternatively, you also have the possibility of selecting a smaller zoom factor, ensuring output to one page ("**Fit to page**" button). Following this, you can enlarge to the original format on a copying machine, to ensure true scaling. Furthermore, you may enter the number of copies to be printed.

If you have activated the **table representation** on the screen, you will see a different dialog box for output by means of the **"File/Print and export"** menu item button **"Printer"**.



Here, you can select the table pages to be printed. In order to achieve output with a zoom factor of 1 (button **"Fit in automatically"** is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in **"File/Print output table"** button **"Output as graphics"**.

- **"DXF file"**
allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- **"GGUCAD file"**
allows output of the graphics to a file, in order to enable further processing with the **GGUCAD** program. Compared to output as a DXF file this has the advantage that no loss of colour quality occurs during export.
- **"Clipboard"**
The graphics are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further processing, e.g. into a word processor. In order to import into any other WINDOWS program you must generally use the **"Edit/Paste"** function of the respective application.
- **"Metafile"**
allows output of the graphics to a file in order to be further processed with third party software. Output is in the standardised EMF format (Enhanced Metafile format). Use of the Metafile format guarantees the best possible quality when transferring graphics.

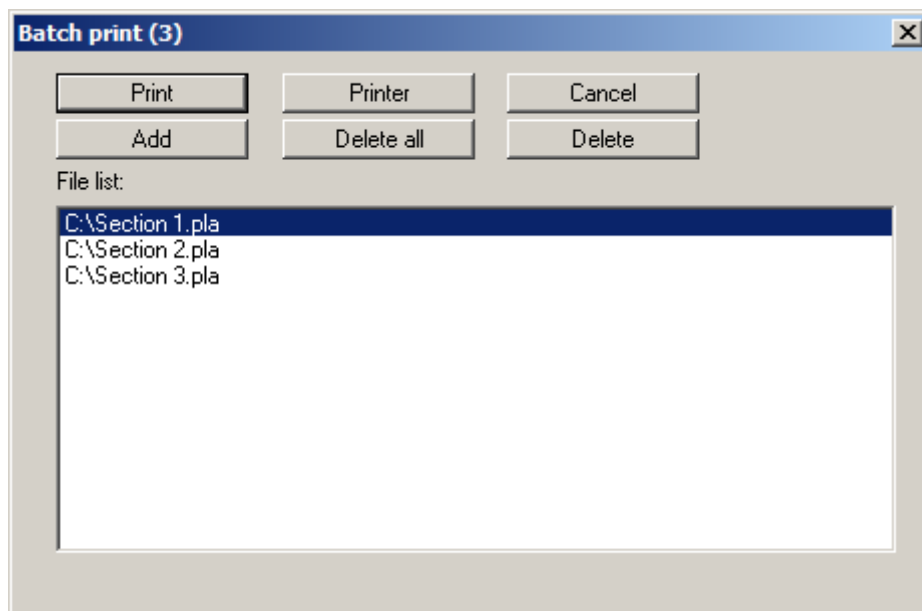
If you select the **"Copy/print area"** tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see **"Tips and tricks"**, Section 8.3).

Using the **"Mini-CAD"** program module you can also import EMF files generated using other GGU applications into your graphics.

- **"MiniCAD"**
allows export of the graphics to a file in order to enable importing to different GGU applications with the **Mini-CAD** module.
- **"GGUMiniCAD"**
allows export of the graphics to a file in order to enable processing in the **GGUMiniCAD** program.
- **"Cancel"**
Printing is cancelled.

7.1.10 "Batch print" menu item

If you would like to print several annexes at once, select this menu item. You will see the following dialog box:



Create a list of files for printing using **"Add"** and selecting the desired files. The number of files is displayed in the dialog box header. Using **"Delete"** you can mark and delete selected individual files from the list. After selecting the **"Delete all"** button, you can compile a new list. Selection of the desired printer and printer preferences is achieved by pressing the **"Printer"** button.

You then start printing by using the **"Print"** button. In the dialog box which then appears you can select further preferences for printer output such as, e.g., the number of copies. These preferences will be applied to all files in the list.

7.1.11 "Exit" menu item

After a confirmation prompt, you can quit the program.

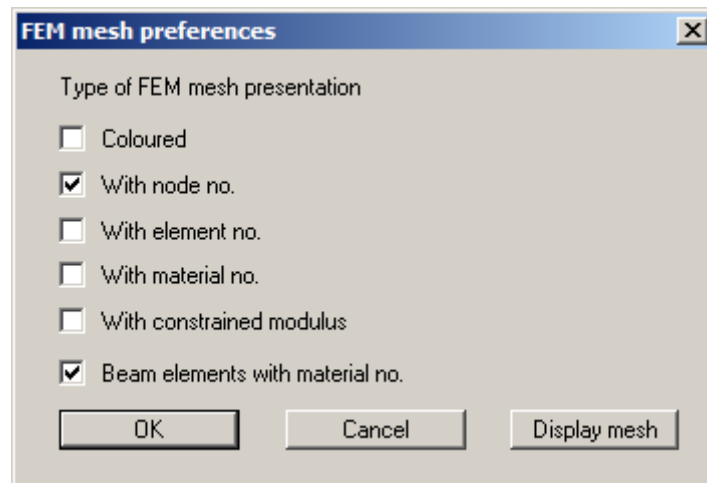
7.1.12 "1, 2, 3, 4" menu items

The **"1, 2, 3, 4"** menu items show the last four files worked on. By selecting one of these menu items the listed file will be loaded. If you have saved files in any other folder than the program folder, you can save yourself the occasionally onerous *rummaging* through various sub-folders.

7.2 FEM mesh menu

7.2.1 "Preferences" menu item

Using this menu item you can define the appearance of the FEM mesh on the screen. The element no. and the material no. cannot be displayed simultaneously.



The "**Display mesh**" button produces a direct representation of the FEM mesh using the selected preferences.

If the FEM mesh representation does not fill the screen, point to the "**Auto-resize**" menu item in the "**Page size + margins**" menu or press [F9].

7.2.2 "FEM mesh" menu item

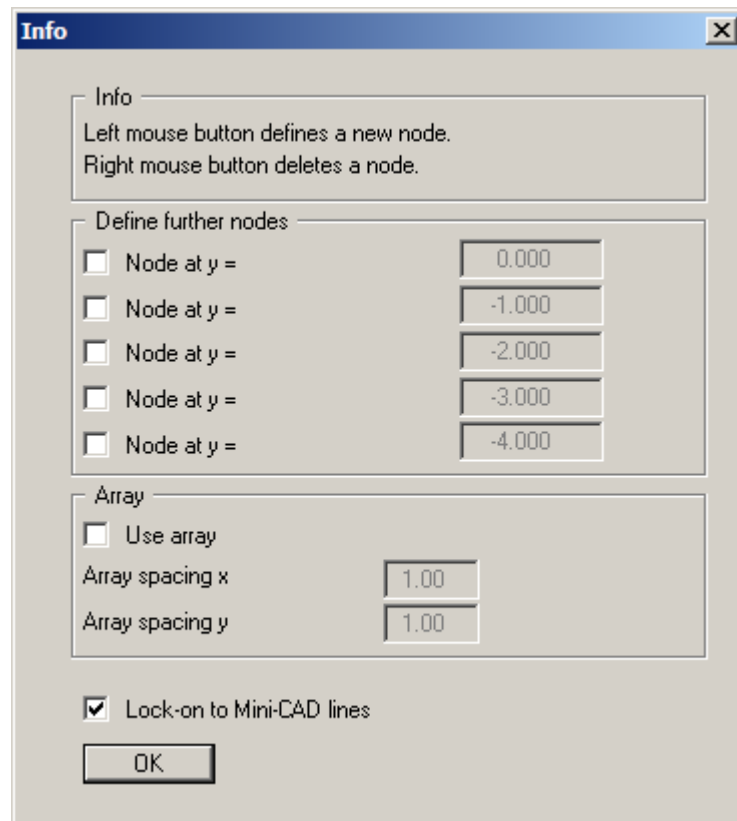
After going to this menu item the FEM mesh is displayed as defined in "**FEM mesh/Preferences**".

7.2.3 "Outline" menu item

After going to this menu item the outlines of the various slab materials used in the FEM mesh are displayed as defined in "**FEM mesh/Preferences**".

7.2.4 "Define (FEM) nodes" menu item

You can use the left mouse button to define a new node or the right mouse button to delete a previously defined node. This menu item can also be reached using [F3]. If you are working a problem using the subgrade reaction modulus method, each new point is assigned the default subgrade (see "Subgrade/Default subgrade" menu item, Section 7.3.2).



When defining nodes the x- and y-coordinates of the current mouse pointer position are shown in the status bar. Using the settings in the "**Define further nodes**" group box in the dialog box it is possible to automatically define up to 5 additional nodes at the same x-coordinate with a single mouse click, but with user-defined y-coordinates. In order to specify the nodes at defined locations, activate the "**Use array**" check box.

If you have access to a scanner you can scan in the system to be processed and save it as a bitmap file (extension: ".bmp"). This bitmap can be displayed on the screen using the **Mini-CAD** program module (see the "**Mini-CAD**" manual). This greatly simplifies input of the principal system nodes. Alternatively, you can also import a DXF file using **Mini-CAD** (see "**Mini-CAD**" manual). This can contain the system outline, for example.

If **Mini-CAD** data are already present, the initial dialog box provides a check box with the option of locking on to **Mini-CAD** lines. If you activate this check box the mouse cursor appears as a rectangle with cross-hairs. If the end point of a **Mini-CAD** line is located within this rectangle the program will lock on to this point precisely; if a number of points are located within the rectangle, it will lock on to the one nearest the centre of the cross-hairs.

7.2.5 "Change (FEM nodes)" menu item

The coordinates of existing nodes can be edited. Three options are available for this:

- **"Via a table"**

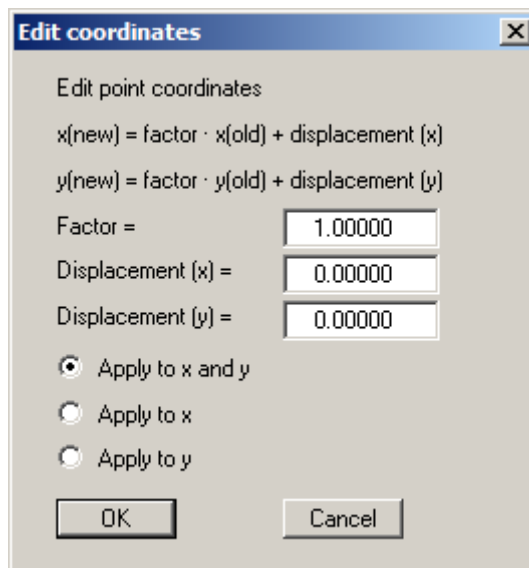
You can edit the coordinates of existing nodes or, alternatively, enter the coordinates of new nodes.

If you need to edit the current number of nodes click the **"x FEM nodes to edit"** button and enter the new number of nodes. You can navigate through the table using **"Forw."** and **"Back"**. If you are working a problem using the *subgrade reaction modulus method*, each new point is assigned the default subgrade (see **"Subgrade/Default subgrade"** menu item, Section 7.3.2). If you set the number of nodes to **0** the FEM incidence table is deleted.

It is even easier to import node coordinates via the Windows clipboard. For example, if the x-/y-coordinates of the FEM mesh nodes are available in an Excel table, it is possible to copy the two columns containing the data into the clipboard (**"Edit/Copy"**) and then to paste them into the dialog box **"Via a table"** by pressing **"Import clipboard"**.

- **"Via equation"**

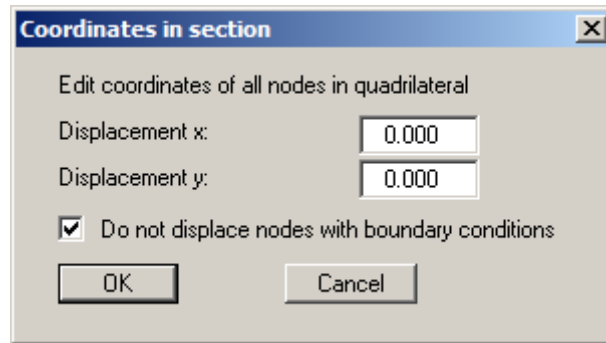
If you have entered the coordinates using the wrong scale, for instance, you can correct this using this menu item.



You also have the option of applying the factor for one direction only. Activate the required option button.

- **"In section"**

The coordinates for a selected area can be displaced by a fixed amount in the x- and the y-direction. Click 4 points counter-clockwise using the mouse. The required displacement can then be entered in the opened dialog box.



If the **"Do not displace nodes with boundary conditions"** check box is activated these points will be excluded from any displacement.

7.2.6 "Move (FEM nodes)" menu item

The defined FEM system is displayed with the finite-elements after selecting this function. The nodes can be moved when holding the left mouse button. The coordinates of the current node are displayed in the status bar. The last node movement can be undone using the [**Back**] key.

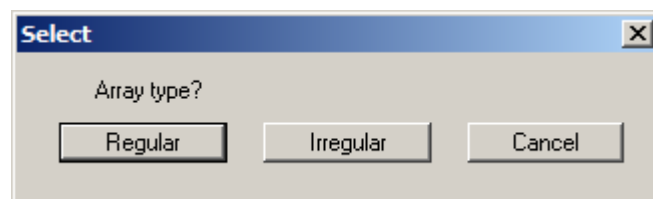
7.2.7 "Edit (FEM nodes)" menu item

By double-clicking a node using the left mouse button a dialog box appears allowing the coordinates to be edited via the keyboard.

7.2.8 "Array" menu item

7.2.8.1 Select type of array

After selecting this menu item you can decide whether new nodes are generated using a regular or an irregular array.



If you are working a problem using the *subgrade reaction modulus method*, each new point generated using this menu item is assigned the default subgrade (see **"Subgrade/Default subgrade"** menu item, Section 7.3.2).

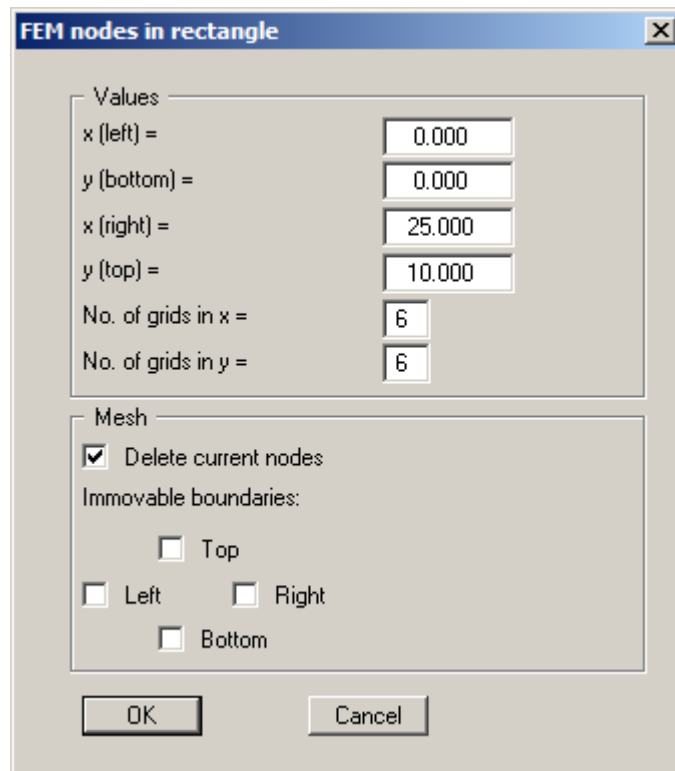
7.2.8.2 Button "Regular"

The regular array allows the nodes to be defined in a number of ways:

- **"Line"** - along one or more lines,
- **"Rectangle"** - in one or more rectangles,
- **"Quadrilateral"** - in one or more quadrilaterals.

The procedure is similar for all three cases. Therefore, only the rectangles will be described.

"Rectangle"



Enter the corner points of the array and the number of subdivisions. If the **"Delete current nodes"** check box is activated, all existing nodes are deleted and then the user-defined FEM nodes in the dialog box are generated with the appropriate FEM mesh. If you want to add further nodes, deactivate the check box.

You can define immovable boundaries directly when generating the FEM mesh. Just activate the check boxes for the required sides. If the check boxes are not activated, all boundaries are generated with a free-earth support. Additional displacement or action boundary conditions can then be defined individually using the **"Boundary"** menu.

If only one of the node points thus generated were to lie within an existing FEM mesh, node generation is aborted with an error message and no further nodes are generated.

7.2.8.3 Button "Irregular"

In contrast to the regular array procedure, where the generated nodes are evenly spaced within the generated rows, the spacing can be varied using the irregular array. This can be done in the following dialog box.

dx values		dy values	
No. of dx values		No. of dy values	
0.000	dx [m]	0.000	dy [m]
1.000	0.5	1.000	1
2.000	0.25	2.000	2
3.000	1.5	3.000	2
4.000	1	4.000	2
5.000	0.5	5.000	1

Recompute x and y values

x0 = 0.00

y0 = 0.00

☒ Generate new mesh

OK Cancel

The array numbers can be defined using the "No. of dx values" and "No. of dy values" buttons. Enter the array spacing in "dx" and "dy". If you press the "Recompute x and y values" button the first column to the left of "dx" and "dy" are recalculated. These are absolute values for the array. The array origin is entered in "x0" and "y0".

If the "Generate new mesh" check box is activated any existing nodes are deleted and a new mesh generated automatically for your newly defined nodes. If the check box remains deactivated existing nodes and the FEM mesh are retained. The new nodes cannot then lie in the same region as the existing FEM mesh. The mesh across the new nodes must then be generated by the user.

7.2.9 "Circular slab, ..." menu item

This menu item makes it simple to define circular slabs and the FEM mesh. It is then possible to select different slabs:

- "Circular slab (simple)"

Create a simple circular slab by means of the diameter, number of rings and segments. A displacement of "0" can be defined at the outer edge. Dimensioning can also be activated in the dialog box. Force and moments can be applied in the slab centre or as distributed loads on a selected ring. The moments can be adopted to calculate the torsional stiffness.

Any existing nodes and FEM mesh are automatically deleted. When you leave the dialog box the program recalculates the coordinates and displays the newly generated circular slab on the screen with the optimum coordinates.

- **"Circular slab (pro)"**
You can generated a more complex circular slab.

Generate circular slab

Geometry

No. of segments: 24

x (centre): 0.000 y (centre): 0.000

☒ Differentiate thickn. [m] d (inside): 1.500 d (outside): 1.500

5 radii to edit Sort radii

Radii [m]

1.000 2.000 3.000 4.000 5.000

Boundary conditions

☐ w = '0' at outer edge

☐ Line load on radius no. 4 Value: 100.0

Force/moments

☒ Use force/moments in slab centre or as distributed load

V [kN]: 10000.000

M(x) [kN·m]: 1000.000

M(y) [kN·m]: 0.000

☐ As distributed load on ring no. 4

☒ Use M(x) and M(y) to calculate torsional stiffness

☐ With dimensions ☒ Delete current nodes

OK Cancel

- **"Annular slab"**
An annular slab can be generated via the number of rings and the internal and external diameters. A displacement of "0" can be defined at the inner and/or outer edges. Force and moments can be applied in the slab centre or as distributed loads on a selected ring. The moments can be adopted to calculate the torsional stiffness.
- **"Annular slab (pro)"**
You can generated a more complex annular slab.
- **"Elliptical slab"**
An elliptical slab can be generated. Simply define the diameters in the x and y directions. A displacement of "0" can be defined in slab centre and/or at the outer edge. Dimensioning can also be activated in the dialog box.

- **"Distributed loads from M+V"**

The distributed loads can be calculated from M and V by the program. The moments can be adopted to calculate the torsional stiffness.

Distributed loads from M and V

Load

V [kN]: 10000.000

M(x) [kN·m]: 1000.000

M(y) [kN·m]: 0.000

☒ Use M(x) and M(y) to calculate torsional stiffness

Info

The program calculates the centroid (xs, ys), the area A and the moments of inertia Ix, Iy and Ixy of the FEM mesh. Together with V, M(x) and M(y), this is used to calculate the stresses at the nodes of the FEM mesh. This provides the distributed loads, which are constant for each element. V, M(x) and M(y) act on the centroid of the FEM mesh.

OK Cancel

The data specified in the message box are then calculated and displayed. They can be copied to the Windows clipboard by pressing **"Copy data to clipboard"**. If the distributed loads are then determined the minimum and maximum stresses are displayed in a message box. After exiting the box by pressing **"OK"** the distributed loads are generated for the entire FEM mesh.

7.2.10 "Manual (FEM) mesh" menu item

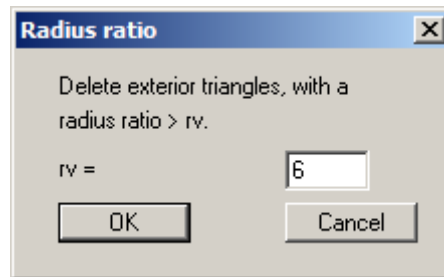
After input of the mesh nodes this menu item is used to define the FEM mesh. Three nodes must be clicked using the left mouse button. Once the three nodes have been selected a box appears for defining the material no. of the FEM element. This menu item can also be reached using **[F4]**. An FEM element can be deleted by selecting the three corresponding nodes once again using the left mouse button.

7.2.11 "Automatic (FEM mesh)" menu item

After entering the mesh nodes automatic mesh generation can be carried out using this menu item (Delauney triangulation). If an FEM mesh already exists it can be either deleted or supplemented. Under certain circumstances, *"air holes"* (incompletely filled FEM mesh) may occur in an existing FEM mesh being supplemented if this mesh was not generated by means of Delauney triangulation. These regions will require manual post-processing or re-triangulation of the complete FEM mesh. All newly generated triangles are assigned the material number **1**.

7.2.12 "Round off (FEM mesh)" menu item

During Delauney triangulation a triangle mesh is generated that envelops all nodes. This can lead to acute-angled triangle elements in the boundary regions. These triangles can be removed from the FEM mesh using this menu item.



The radius ratio describes the relationship between external radius and internal radius of a triangle. For an equilateral triangle, this ratio equals 2.0 (optimum). Before you arrive at the dialog box above you will be shown the maximum and minimum radius ratios. In the example above, all external triangles with a radius ratio greater than 6 will be removed. To avoid chaos, only exterior triangles will be removed.

7.2.13 "Delete (FEM mesh)" menu item

This menu item allows you to delete selected system triangles. You must first click four points in anti-clockwise direction. All triangles with their centroid within this quadrilateral will be deleted.

7.2.14 "Optimize (FEM mesh)" menu item

You first select in a dialog box whether the diagonals or the topology should be optimised.

7.2.14.1 "Diagonals" button

Optimisation of diagonals is implemented in order to create a numerically favourable FEM mesh, i.e. where possible, equilateral triangles. The effect of optimisation of diagonals can best be seen using an example:

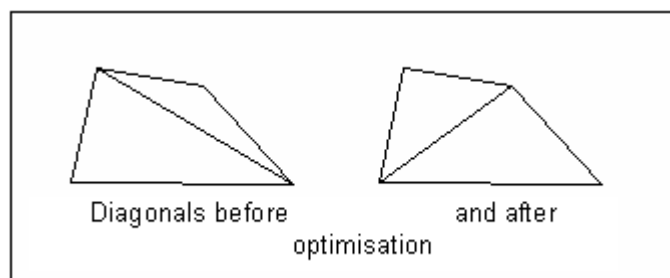


Figure 1 Optimisation of diagonals

If an existing *unfavourable* diagonal cuts two different material areas, no optimisation takes place because this would alter the system.

7.2.14.2 "Topology" button

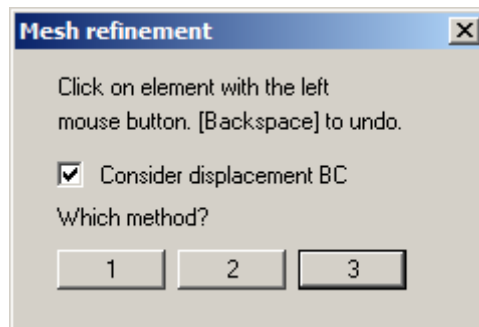
This program routine displaces the triangle element nodes so that, where possible, equilateral triangles are created. Equilateral triangles have especially favourable numerical properties. Because the displacement of system boundaries and element boundaries with neighbouring elements consisting of different materials does not make much sense, these element boundaries are fixed from the outset. Nodes with defined displacement boundary conditions also remain unaltered. Optimisation of the FEM mesh can be followed on the screen by setting the "**With graphics**" check box. The optimisation routine can be aborted at any time by pressing the right mouse button.

7.2.15 "Align (FEM mesh)" menu item

Select a node by clicking the left mouse button. The node can then be aligned on a circle, for example.

7.2.16 "Refine individually" menu item

Mesh elements can be selected for refinement using the following menu item.



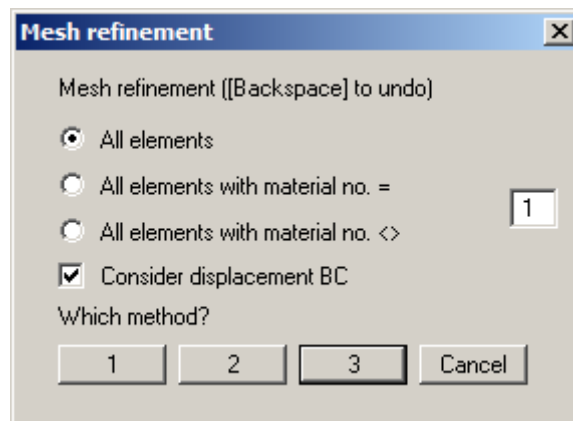
Upon activating the "**Consider displacement BC**" check box, new nodes located immediately between two nodes with displacement boundary conditions will be assigned the average of the two values. This procedure is not unequivocal when applied to *action boundary conditions* and can lead to misunderstandings. Action boundary conditions are therefore not refined in the course of mesh refinement. A description of the 3 refinement methods can be found in Section 7.2.19.

7.2.17 "Section" menu item

A number of elements previously enveloped in a polygon can be refined using this menu item. Displacement boundary conditions can be taken into consideration (see "**FEM mesh/Refine individually**" menu item, Section 7.2.16). A description of the 3 refinement methods can be found in Section 7.2.19.

7.2.18 "All" menu item

The following dialog box appears after selecting this menu item or, alternatively, pressing [F7]:



Either all elements or only element with certain material numbers can be refined. Here, too, displacement boundary conditions can be taken into consideration (see " **FEM mesh/Refine individually**" menu item, Section 7.2.16). A description of the 3 refinement methods can be found in the following Section 7.2.19.

7.2.19 Refinement methods

Three different refinement methods can be applied for element refinement. Refinement will be demonstrated on the following mesh using element 23 as an example.

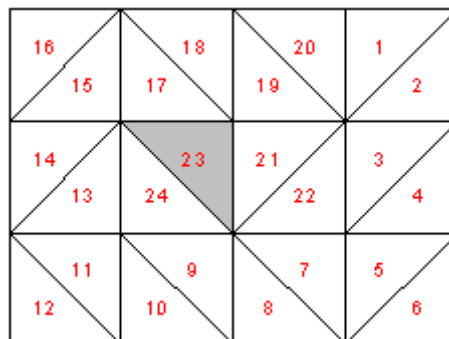


Figure 2 FEM refinement demonstration mesh

- **"Method 1"**

An additional node is generated in the centroid of the selected triangle.

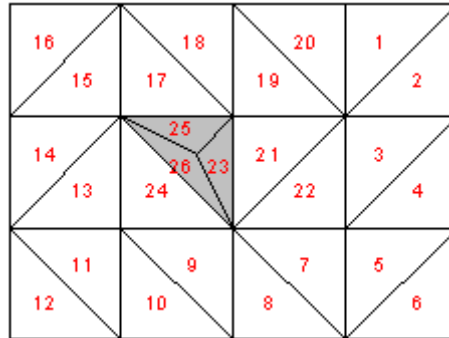


Figure 3 FEM mesh refinement using Method 1

- **"Method 2"**

The selected triangle element and the neighbouring triangle element are halved.

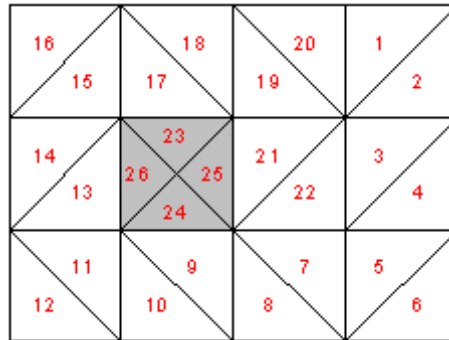


Figure 4 FEM mesh refinement using Method 2

- **"Method 3"**

A new triangle element is inserted at the median of the clicked triangle element. The neighbouring triangle elements are halved.

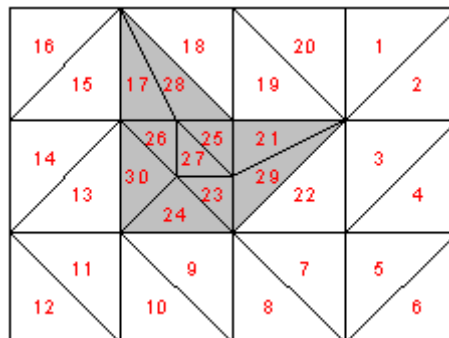


Figure 5 FEM mesh refinement using Method 3

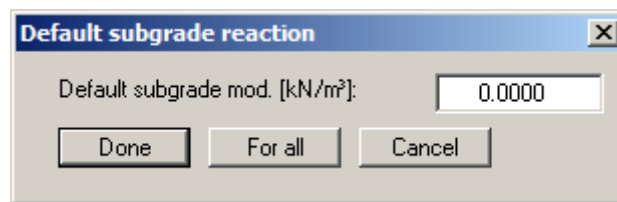
7.3 Subgrade menu (subgrade reaction modulus only)

7.3.1 General note on subgrade reaction moduli

The "**Subgrade**" menu is only visible if the subgrade reaction modulus method is selected in "**File/New**". This menu enables simple definition of the subgrade reaction moduli assigned to each mesh node.

7.3.2 "Default subgrade" menu item

The default subgrade reaction modulus defined in this dialog box is assigned to all new FEM nodes defined using the "**FEM mesh/Define nodes**" menu item.



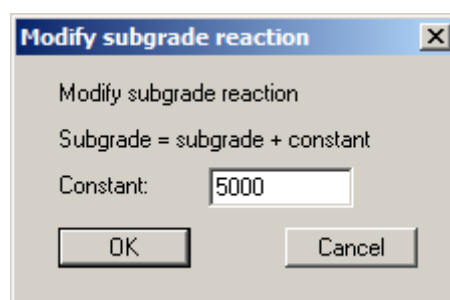
If you edit the given value and leave the dialog box using "**Done**" the newly defined subgrade reaction modulus will be adopted as of this moment for newly defined nodes. If nodes are already defined, they can also be retroactively assigned a default subgrade reaction modulus using the "**For all**" button.

7.3.3 "Individual subgrade" menu item

You can edit the subgrade reaction modulus by double-clicking a node. In the opened dialog box you also have the option of editing the coordinates of the selected node.

7.3.4 "Modify (subgr. reac. mod. at FEM nodes)" menu item

You can modify all subgrade reaction moduli as a function of the existing subgrade reaction moduli. In the example shown in the following dialog box the subgrade reaction modulus at all nodes was increased by 5000 kN/m³.

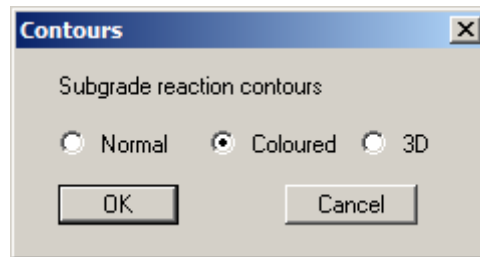


7.3.5 "In section" menu item

After defining a quadrilateral, all nodes within it can be assigned a new subgrade reaction modulus. If the "**Display subgrade reaction**" check box is activated in the message box the nodes are labelled with the current subgrade reaction modulus. This provides a good overview of the possible subgrade reaction modulus ranges present.

7.3.6 "Subgrade contours" menu item

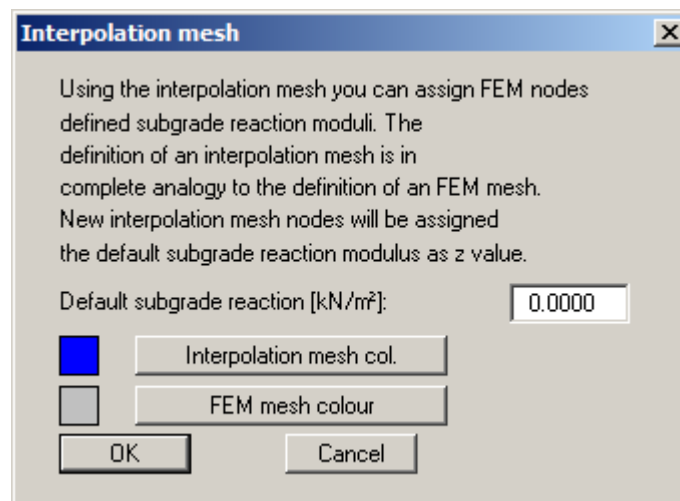
In order to better visualise the subgrade reaction modulus situation you can generate a contour diagram of the defined subgrade reaction moduli.



Define the type of contour diagram using the "**Normal**", "**Coloured**" and "**3D**" option buttons. Details are described in Sections 7.9.2 to 7.9.4.

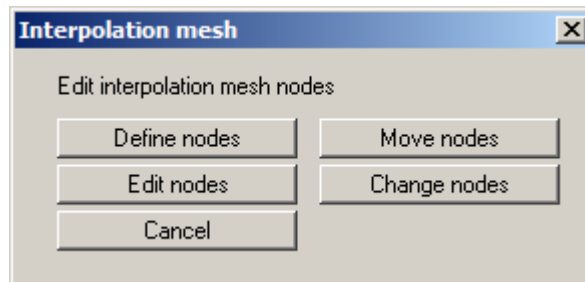
7.3.7 "Interpolation mesh" menu item

In principle, the subgrade reaction moduli can be completely defined using the previous menu items. A further defining simplification is offered by the interpolation mesh. Similarly to the FEM mesh, this interpolation mesh consists of nodes and triangles. Certain subgrade reaction moduli can be defined at the nodes. After defining an interpolation mesh you can assign the thus defined subgrade reaction modulus profile to the underlying FEM mesh. These are assigned by means of linear interpolation. The interpolation mesh should (but not must) completely cover the FEM mesh. In the simplest case possible, this may consist of one triangle.



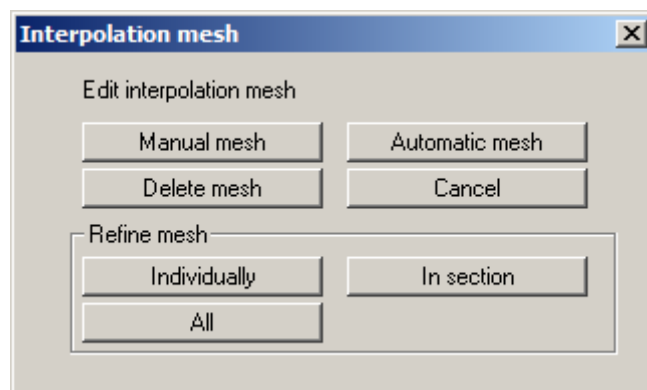
7.3.8 "Nodes" menu item

In complete analogy to the FEM mesh (see Sections 7.2.4 bis 7.2.7) you can define, move, edit or change the interpolation mesh nodes.



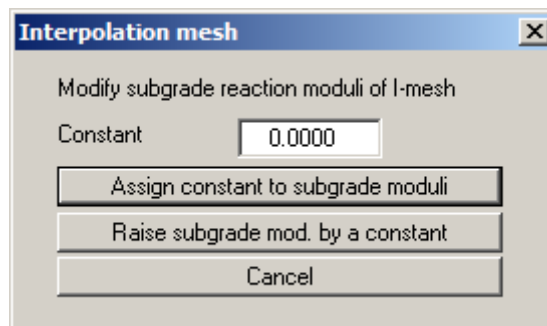
7.3.9 "Mesh" menu item

In complete analogy to the FEM mesh (see Sections 7.2.10, 7.2.11, 7.2.13 und 7.2.16 bis 7.2.18) you can edit the interpolation mesh.



7.3.10 "Modify" menu item

The subgrade reaction moduli of given interpolation mesh nodes can be modified. After clockwise definition of a quadrilateral the following dialog box appears:



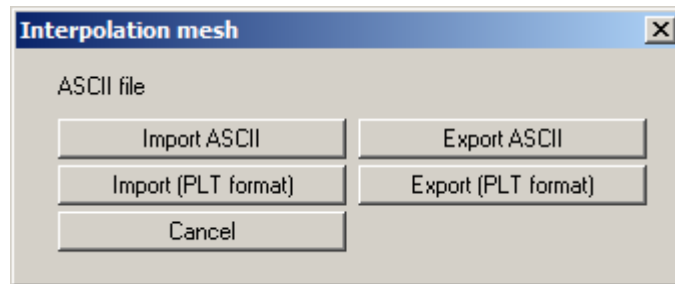
Once a constant has been entered the two options shown in the dialog box are available for modifying the selected subgrade reaction moduli.

7.3.11 "Contours" menu item

You can create a contour diagram of the interpolation mesh subgrade reaction moduli. Either coloured contours or a 3D representation can be selected (also see the "**Subgrade/Contours**" menu item, Section 7.3.5).

7.3.12 "Import/export" menu item

The x- and y-coordinates of the interpolation mesh can be imported from an ASCII file.



If this is done in PLT format, the triangle and subgrade reaction moduli information will also be exported or imported. A dialog box as described in Section 7.1.5 opens for the "**Import ASCII**" option.

7.3.13 "Assign" menu item

After completely processing the interpolation mesh the subgrade reaction moduli associated with the interpolation mesh can be assigned to the FEM mesh.

Because no undo is possible after assigning the subgrade reaction moduli, it is expedient to save the file beforehand.

7.4 Constrained mod. menu (constrained modulus method only)

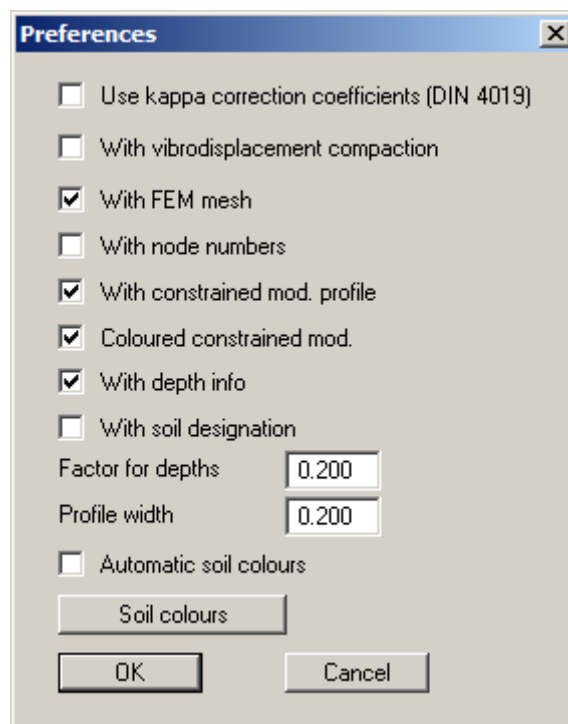
7.4.1 General note on constrained moduli

The "**Constrained mod.**" menu is only visible if the constrained modulus method is selected in "**File/New**". This menu serves simple definition of the constrained modulus.

It is necessary to define a triangle mesh for analysis using the constrained modulus method. This mesh is not related to the FEM mesh. The nodes of this triangle mesh are described by the drilling points. The drilling points are then connected to form a triangle mesh, from which the program can interpolate the constrained modulus profile at any point. The constrained modulus profiles are described by a mesh, structured similarly to the interpolation mesh (see the "**Subgrade**" menu). The difference is that the subgrade reaction moduli are not defined at the nodes of this mesh, but a constrained modulus profile. In the simplest case this mesh can consist of a single triangle comprising three constrained modulus profiles.

7.4.2 "Preferences" menu item

Using this menu item general presentation parameters can be given, thus influencing the system graphics.



DIN 4019 Table 1 allows correction coefficients kappa to be taken into consideration for modelling settlements. These correction coefficients can be activated using the appropriate check box in the dialog box.

Activate vibrodisplacement compaction adoption after Priebe (Heinz J. Priebe, *Die Bemessung von Rüttelstopfverdichtung*, Ground Engineering, December 1995) using the "**With vibrodisplacement compaction**" check box.

"**Factor for depths**" allows you to control the length of the profile columns in the presentation. If the "**Automatic soil colours**" check box is activated, the profiles will be automatically colour-filled by the program. Otherwise, the colours defined in "**Soil colours**" will be used for the individual soil layers. The selected colour setting is automatically adopted for the constrained modulus legend (see Section 7.7.10).

7.4.3 "Layers" menu item

After clicking this menu item the following dialog box appears:

No	Es [MN/m²]	gamma [kN/m³]	nu [-]	Designation
1	1.00	14.000	0.000	Clay
2	20.00	9.000	0.000	Sand, silty

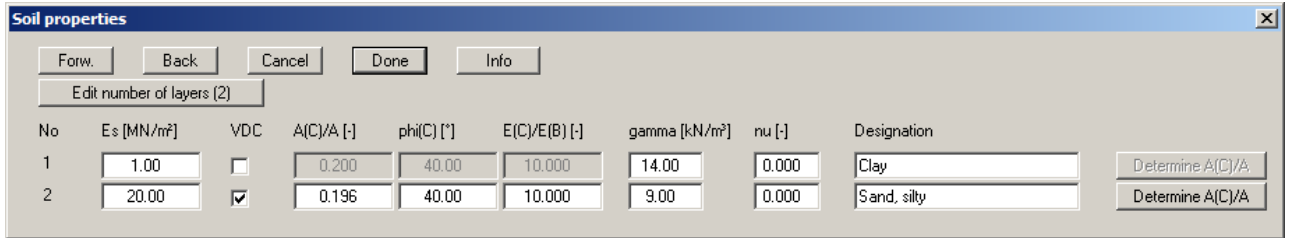
In the dialog box you can enter the names, constrained moduli and Poisson's ratios ν of the individual layers. Input of the unit weight of the wet soil is necessary for the limiting depth analysis, or the buoyant unit weight for soils below the groundwater table. This input box only appears if the "**With limiting depth analysis**" check box is activated in "**File/New**" or "**System/Analyse**".

If the constrained modulus is used for analysis, ν equals 0.0 for all soils. The value of ν displayed in the constrained mod. legend can then be switched off (see Section 7.7.10).

If you need to edit the number of layers, select the "**Edit no. of layers**" button and then enter the new number of layers. Edit the soil properties if required.

In order to ensure a sensible interpolation within the triangle mesh it is necessary that all nodes possess the same number of layers. If your system contains areas in which certain soil layers do not occur (e.g. peat lenses only in certain areas), then simply assign these layers a thickness of "**0.0**" at the appropriate nodes, using the "**Default depths**" menu item (see Section 7.4.4), i.e. a height corresponding to the layer above.

If you are working with vibrodisplacement compaction the following dialog box opens after clicking this menu item:



The "Soil properties" dialog box contains the following controls and data:

Buttons: Forw., Back, Cancel, Done, Info

Text field: Edit number of layers (2)

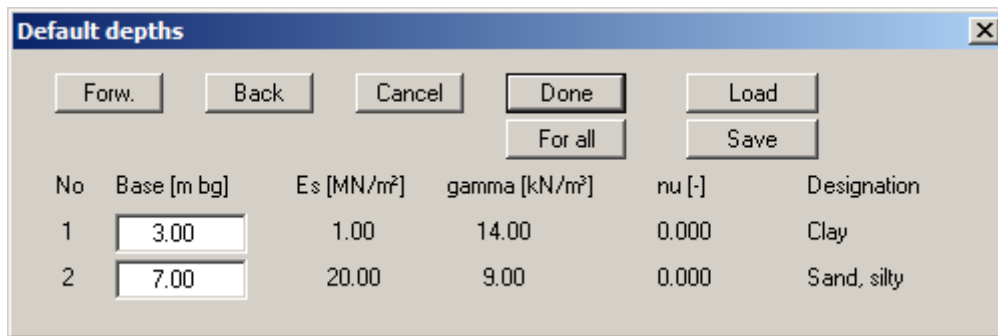
No	E_s [MN/m ²]	VDC	$A(C)/A$ [-]	$\phi(C)$ [°]	$E(C)/E(B)$ [-]	γ [kN/m ³]	ν [-]	Designation
1	1.00	<input type="checkbox"/>	0.200	40.00	10.000	14.00	0.000	Clay
2	20.00	<input checked="" type="checkbox"/>	0.196	40.00	10.000	9.00	0.000	Sand, silty

Buttons on the right: Determine $A(C)/A$, Determine $A(C)/A$

Define the soil layer improved by vibrodisplacement compaction by activating the "VDC" check box. Enter the corresponding parameters for this layer (see "Info" button). The ratio of the areas " $A(C)/A$ " can be determined and adopted in the table by pressing the far right button. The ratio " $E(C)/E(B)$ " should not be greater than 15 to a maximum of 20.

7.4.4 "Default depths" menu item

After clicking this menu item the following dialog box appears:



The "Default depths" dialog box contains the following controls and data:

Buttons: Forw., Back, Cancel, Done, Load, For all, Save

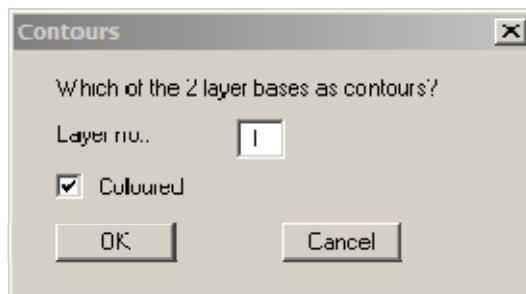
No	Base [m bg]	E_s [MN/m ²]	γ [kN/m ³]	ν [-]	Designation
1	3.00	1.00	14.00	0.000	Clay
2	7.00	20.00	9.00	0.000	Sand, silty

If you have entered two layers using the "Constrained mod./Layers" menu item, two layers will also be shown in this dialog box. The corresponding constrained moduli and the Poisson's ratio are also shown for your information. The given depths can be edited to suit your wishes. Depths are entered as m below grade. During the subsequent definition of triangle nodes, these layer depths are assigned to the corresponding nodes. The layer depths for each node can be edited as required. If the constrained modulus profile is equal for almost all nodes, you can save input work during the initial definition of default depths.

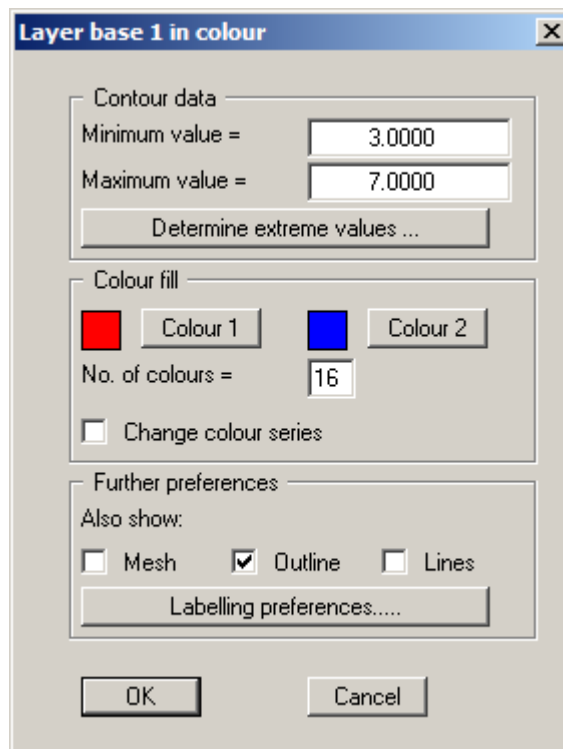
A previously saved depth distribution can be loaded using the "Load" button. Using "Save" you can save the current depth distribution to a file in order to have it available later without requiring renewed input. If you press the "For all" button all current triangle nodes will be assigned these depths. In the dialog box a maximum of 8 layers are displayed. If more than eight layers are present you can navigate through the table using the "Forw." and "Back" buttons.

7.4.5 "Layer base contours" menu item

After selecting this menu item the following dialog box appears:



This menu item serves as a check for the layer depth input data. Enter the number of the layer for which you would like to see a contour diagram of the base. If you activate the "**Coloured**" check box you will see a colour filled contour diagram. Otherwise a normal contour of diagram will be drawn (also see Section 7.9.2).



For colour-filled graphics you will see the above dialog box.

- "**Contour data**" group box
Press the "**Determine extreme values**" button. The program then determines the minimum and maximum values for the corresponding layer base. You can then edit these values, for example in order achieve a defined start value.

- **"Colour fill"** group box
You can control the colour subdivisions of the contour diagram using **"No. of colours"**. In the example above, 16 colours will be displayed between **"Colour 1"** and **"Colour 2"**. The default setting is a colour course from red to blue. These colours can be edited as required after selecting the **"Colour 1"** and **"Colour 2"** buttons, or simply reverse the choice by selecting the **"Change colour series"** check box.
- **"Further preferences"** group box
In addition to the colour presentation you can also have the triangle mesh and/or the outline displayed. Additional contour lines can also be drawn. Line labelling preferences can be defined by means of the **"Labelling preferences"** button.
- **"OK"**
The colours will be drawn after confirmation. A colour bar at the right edge of the sheet allows correlation between the colour and the corresponding value. If this colour bar is drawn in the right page margin, specify a larger value for the right plotting margin (e.g. 25 mm) in the **"Page size + margins/Page size and margins"** menu item (see Section 7.8.4).

7.4.6 "Edit layer depths" menu item

This menu item allows editing of the base of a layer at the individual nodes of the triangle mesh. After double-clicking near the node to be edited you can change the depth of the layer base.

No	Base [m bg]	Es [MN/m ²]	gamma [kN/m ³]	nu [-]	Designation
1	2.00	1.00	14.00	0.000	Clay
2	4.00	20.00	9.00	0.000	Sand, silty

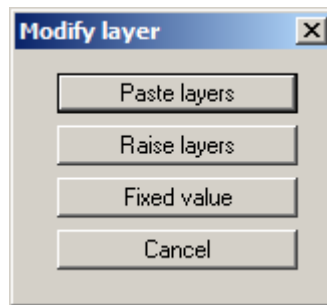
7.4.7 "Determine" menu item

Have the constrained modulus profile displayed for any point within the triangle mesh.

No	Base [m bg]	Es [MN/m ²]	gamma [kN/m ³]	nu [-]
1	5.33	1.00	14.000	0.000
2	6.66	20.00	9.000	0.000

7.4.8 "Modify" menu item

After clicking this menu item the following dialog box appears:



- **"Paste layers"**
A new layer can be defined by dividing an existing layer in all constrained modulus profiles. The depth range of the selected layer is divided between 2 new layers. The final depth of the constrained modulus profile thus remains neutral. The soil properties of the selected layer will be adopted for both layers.
- **"Raise layers"**
A layer can be selected and raised or lowered by a given amount. This action applies to all constrained modulus profiles.
- **"Fixed value"**
A selected layer can be retroactively assigned a given depth in all constrained modulus profiles.

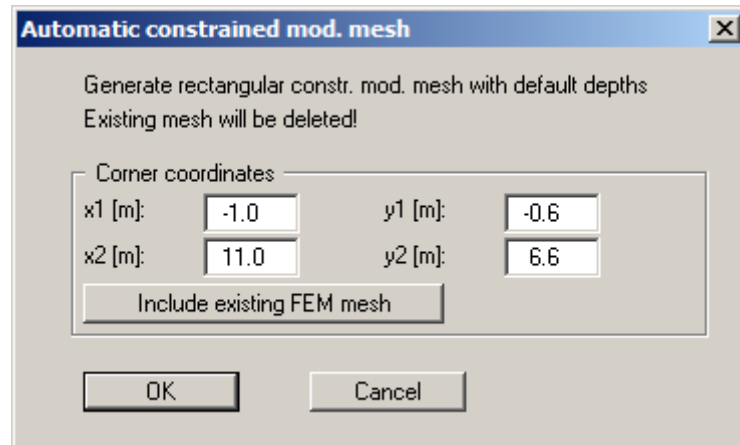
7.4.9 "Display in section" menu item

A section can be drawn at any location through the constrained modulus profile mesh and the applicable geological section displayed. This procedure corresponds to the section representation described in the **"Evaluation/Any section"** menu item (see Section 7.9.10).

7.4.10 "Automesh" menu item

If this menu item is selected a rectangular constrained modulus mesh with the default depths can be generated. You need only enter the corner coordinates.

But beware: The existing constrained modulus mesh is deleted!



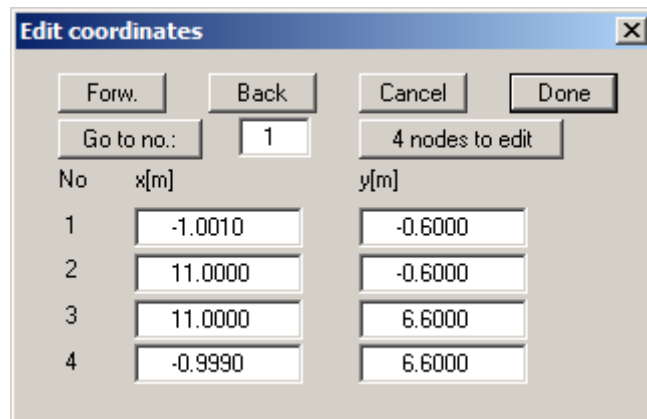
If you first click the **"Include existing FEM mesh"** button the program automatically determines the required coordinates around the FEM mesh and enters them into the input boxes.

7.4.11 "Define nodes" menu item

Using this menu item the triangle nodes for the constrained modulus profile mesh can be defined by mouse click. Nodes are defined using the left mouse button and deleted using the right mouse button. In terms of the constrained modulus profile the new nodes are allocated to the layers with the standard depths, as previously defined using **"Constrained mod./Layers"** and **"Constrained mod./Default depths"** menu items (see Sections 7.4.3 and 7.4.4).

7.4.12 "Change" menu item

After selection of this menu item, the following dialog box appears:



The "Edit coordinates" dialog box features a title bar with a close button. It contains four buttons at the top: "Forw.", "Back", "Cancel", and "Done". Below these is a "Go to no.:" label followed by a text box containing the number "1". To the right of this is a button labeled "4 nodes to edit". The main area of the dialog is a table with three columns: "No", "x[m]", and "y[m]". It lists four nodes with their respective coordinates.

No	x[m]	y[m]
1	-1.0010	-0.6000
2	11.0000	-0.6000
3	11.0000	6.6000
4	-0.9990	6.6000

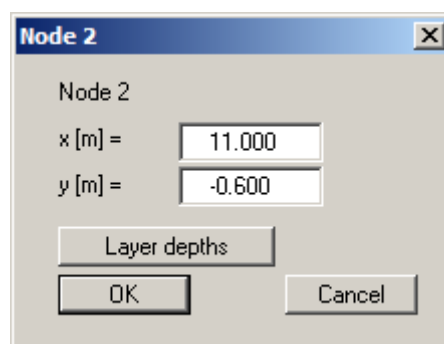
In this box it is possible to change or enter the x- and y-coordinates of the nodes (drilling points) using the keyboard. If you increase the number of nodes using the "**x nodes to edit**" button the new nodes are assigned the default depths with regard to the constrained modulus profile. The button operations correspond to those in the "**Constrained mod./Layers**" menu item (see Section 7.4.3).

7.4.13 "Move" menu item

After clicking this menu item it is possible to move current triangle nodes using the mouse and thus alter the x- and y-coordinates.

7.4.14 "Edit" menu item

Using this menu item it is possible to retroactively edit individual nodes of the constrained modulus profile mesh. By double-clicking near a node the following dialog box opens, allowing the coordinates to be edited.



The "Node 2" dialog box has a title bar with a close button. It displays "Node 2" and shows the x and y coordinates for that node. The x coordinate is 11.000 and the y coordinate is -0.600. Below these fields is a button labeled "Layer depths". At the bottom are "OK" and "Cancel" buttons.

Node 2
x [m] = 11.000
y [m] = -0.600
Layer depths
OK Cancel

After clicking the "**Layer depths**" button a further dialog box opens for editing the layer depths.

7.4.15 "Manual mesh" menu item

You can *manually* combine triangle nodes to a triangle mesh. Normally, you would have the triangulation (see below) carried out automatically. However, if the triangle mesh generated automatically by the program does not suit your requirements you can carry out the triangulation *manually*. If you need to delete an existing triangle, click the nodes of this triangle again.

7.4.16 "Automatic" menu item

The "Automatic" menu item runs a program controlled mesh generation routine (triangulation). You have the option of deleting or supplementing an existing triangle mesh. You should Only select "Supplement" in exceptional cases, as triangulation follows certain laws which may not allow sensible complementing of an existing current partial mesh.

7.4.17 "Delete" menu item

You can delete the complete constrained modulus profile mesh after a security request.

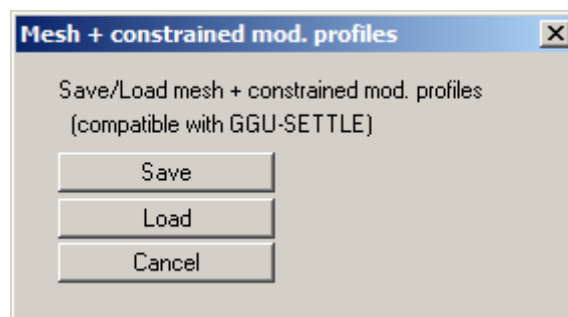
7.4.18 "Refine" menu item

The program allows the refinement of an existing triangle mesh. Refinement is generally only necessary if you need to supplement an existing triangle mesh with further, possibly fictitious, nodes.

Refinement of the constrained modulus profile mesh is in analogy to the refinement of individual elements of the FEM mesh (see Section 7.2.16).

7.4.19 "Save/Load" menu item

If you are analysing slabs using the constrained modulus method, you can save your constrained modulus profiles with the corresponding mesh in a file with the ".**ggu_stei**" extension using this menu item, or open a previously saved file.



These files are compatible with the **GGU-SETTLE** program, meaning that the constrained modulus profiles and the mesh can be imported into **GGU-SETTLE** for a subsequent settlement analysis.

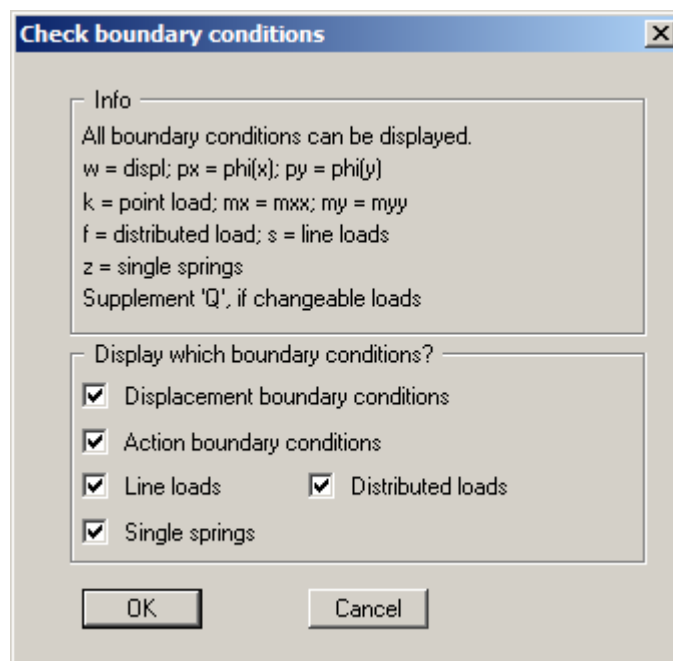
7.5 Boundary menu

7.5.1 "Preferences" menu item

In the dialog box for this menu item preferences for the presentation of the boundary conditions on the screen can be specified.

7.5.2 "Check" menu item

This menu item provides a good overview of the locations of all boundary conditions by displaying the required boundary conditions on the screen.



7.5.3 "Single displacement + load" menu item

Using this menu item it is possible to allocate displacement boundary conditions or action boundary conditions to individual FEM nodes and to modify or delete existing boundary conditions. Left-click the required node to open the following dialog box:

Node: 33 (x / y = 3.0000 / 2.4000)

Displacement boundary conditions

☐ Displacement (positive upwards) 0.0000

☐ ϕ (x) (positive clockwise around x-axis) 0.0000

☐ ϕ (y) (positive clockwise around y-axis) 0.0000

Action boundary conditions

☒ Point load (positive downwards) 100.0000 ☐ Live

☐ M (x) (positive clockwise around x-axis) 0.0000 ☐ Live

☐ M (y) (positive clockwise around y-axis) 0.0000 ☐ Live

OK Cancel

Activate the check box in front of the required boundary condition and enter the corresponding value. A displacement or a rotation in the x- or y-direction can be defined for a displacement boundary condition. Deactivate the corresponding check box to delete a boundary condition.

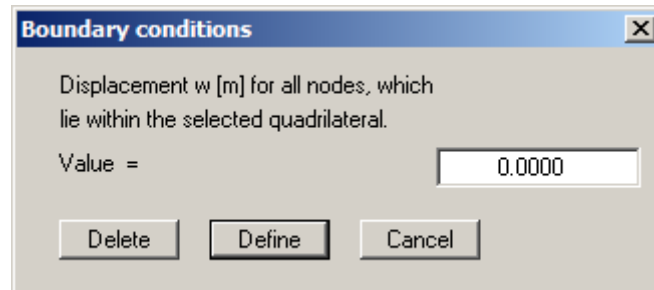
The point loads are defined within the FEM system in analogy to the displacement boundary conditions. The action boundary conditions can be defined as Point load and Moment M (x) and Moment M (y). When analysing with the new *partial safety factors* you will also see the "Live" check box because of the differentiation between permanent and changeable surcharges.

Note on free-earth support:

The case of a free-earth support is automatically considered by the finite-element-method. All system boundaries or partial system boundaries possessing no action or displacement boundary conditions automatically have a free-earth support.

7.5.4 "(Displacement) In section" menu item

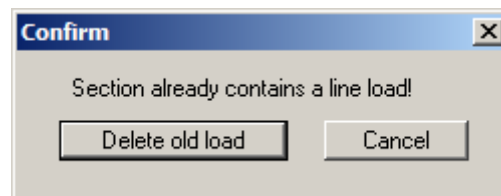
After selecting this menu item several FEM nodes can be simultaneously assigned a displacement boundary condition in a defined section, or an existing boundary condition be deleted. First select the type of displacement boundary condition. Then click four points in an anti-clockwise direction using the left mouse button. The right mouse button will undo. The definition refers to all nodes within the quadrilateral. The desired action can then be selected in a dialog box..



7.5.5 "Line loads" menu item

The procedure for defining line loads is explained in a dialog box. Define the line load by clicking the system nodes to which a boundary condition is then assigned. The action is completed by pressing [**Return**]. A dialog box into which you can enter the start and end value then opens. The load is then assigned corresponding to the point distance of the neighbouring element edges. When analysing with the new *partial safety factors* you will also see the "**Live**" check box because of the differentiation between permanent and changeable surcharges.

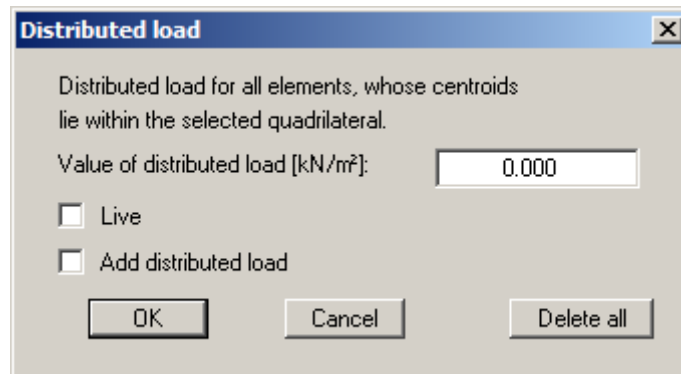
If line loads have already been defined an additional "**Delete all line loads**" button appears in the dialog box. If you leave the dialog box by using this button, all line loads are deleted and a new line load can be defined. Individual line loads can be deleted by pointing to this menu item once again and clicking the nodes to which the line load was assigned. The following dialog box then appears:



Click the "**Delete old load**" until all sections of the marked line load are deleted and then define the new line load if required.

7.5.6 "Distributed loads" menu item

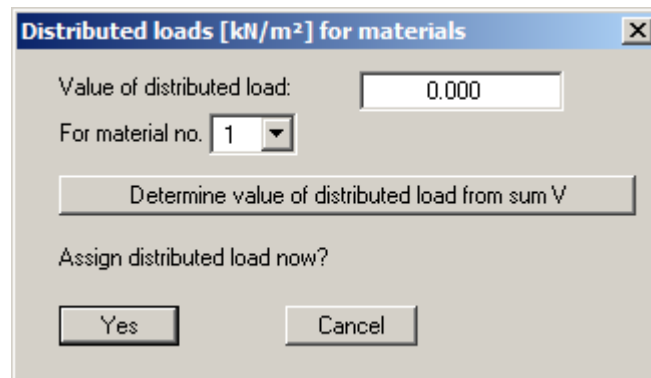
Certain FEM elements can be assigned distributed loads by defining the area required as a polygon using 4 counter-clockwise points. You will then see the following dialog box:



When analysing with the new *partial safety factors* you will also see the "Live" check box because of the differentiation between permanent and changeable surcharges. If you activate the "Add distributed load" check box, the value entered will be added to any distributed loads already present. If negative entries are made the distributed load is reduced correspondingly. If you click the "Delete all" button, all existing distributed loads are deleted and you can define a new distributed load after specifying a new polygon.

7.5.7 "Distributed loads for materials" menu item

Certain materials can be assigned distributed loads. You will see the following dialog box:



The size of the distributed load can be entered directly or determined by the program by pressing the "Determine value of distributed load from sum V" button. Sum V is then entered in an additional dialog box. The program calculates the distributed load using V/A and the area A determined for the selected material number.

7.5.8 "Individual materials" menu item

After selecting the type of FEM system screen representation the material number assignments can be edited for individual elements of the FEM mesh. This assigns these elements the material properties that were defined in the "**System/Slab material**" menu item (see Section 7.6.3).

7.5.9 "(Materials) In section" menu item

After selecting the type of FEM system screen representation you define a polygon by clicking 4 points in counter-clockwise direction. All elements with their centroid within this polygon can be assigned a different material number. The associated material properties are defined in the "**System/Slab material**" menu item (see Section 7.6.3).

7.5.10 "Beams" menu item

After you have defined an FEM mesh you can define beams along nodes of the FEM mesh. Their stiffness can later be taken into consideration during the slab calculations. This can be useful when taking the influence of footing beams or similar elements into account, for example. In principle, it is also possible to model a footing beam by thickening the slab in the appropriate areas.

After completing node selection by pressing the **[Return]** key a dialog box opens for assigning the defined beam a material number. If there are several footing beams in the system, with differing thicknesses and/or widths, for example, you can assign a different material number for each type.

The defined beams are shown with the appropriate material number in the graphics. The colour and width of the lines can be edited using the "**Graphics preferences/Pen colour and width**" menu item (see Section 7.7.3) .

If you need to delete certain beams, click on these beams once again. Individual sections of the defined beam are deleted using the "**Delete old beam**" button, i.e. *some* of the defined beam may remain.

7.5.11 "Delete all" menu item

All defined beams are deleted after a prompt.

7.5.12 "Single springs" menu item

Beside beam elements, individual system nodes can also be assigned single springs. The application of single springs may be required if you need to investigate a slab, for example, resting on only slightly load-bearing ground and which is therefore supported in various locations by piles.

After clicking a node a dialog box opens for defining the magnitude of the single spring in kN/m. Input ≤ 0.0 delete an existing spring.

The size of the single spring is acquired from the resistance-settlement diagram to DIN 4014, for example. For instance, if a pile displays a settlement of $s = 0.95$ cm with a bearing capacity of $P = 1340$ kN, the single spring size C_B is calculated from:

$$C_B = P/s = 1340/0.0095 = 141,000.0 \text{ kN/m}$$

If you also need to take the axial stiffness of the pile into consideration the spring constant of the pile C_P must be determined from:

$$C_P = E \cdot A/L$$

E = Young's modulus of pile
 A = cross-sectional area of pile
 L = length of pile

Example:

$$E = 30,000,000 \text{ kN/m}^2 \text{ (concrete)}$$

$$A = 0.20 \text{ m}^2$$

$$L = 12.0 \text{ m}$$

Giving

$$C_P = 30,000,000 \cdot 0.20/12.0 = 500,000.0 \text{ kN/m}$$

The size of the single spring (overall spring constant) C is acquired from:

$$1/C = 1/C_B + 1/C_P$$

$$1/C = 1/141,000 + 1/500,000 = 9.0922 \cdot 10^{-6}$$

$$C = 109,984 \text{ kN/m}$$

7.6 System menu

7.6.1 "Info" menu item

You will see information on the current system with the number of elements, nodes and boundary conditions in a dialog box.

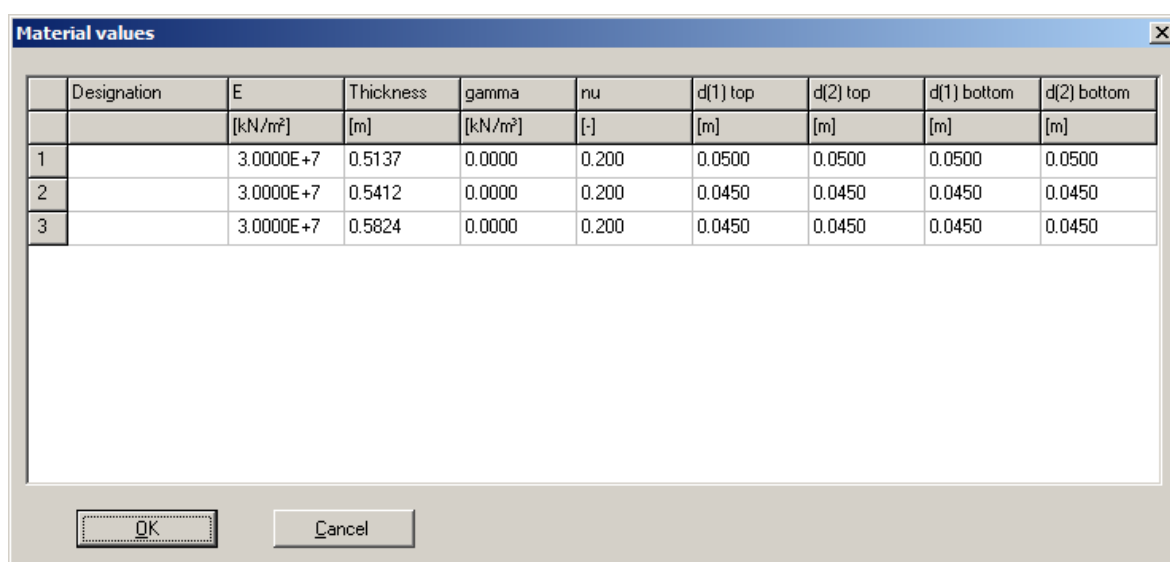
7.6.2 "Units + project identification" menu item

You can enter a description of the current system; this will then be used in the *General legend* (see Section 7.7.8).

Select the units for the constrained modulus and the displacement in the second input line.

7.6.3 "Slab material" menu item

The slab parameters can be edited. Depending on the number of material numbers assigned using the "**Boundary/Individual materials**" (Section 7.5.8) or "**Boundary/(Materials) In section**" (Section 7.5.9) menu items, a dialog box opens with the corresponding number of input lines:



The dialog box titled "Material values" contains a table with 10 columns: Designation, E, Thickness, gamma, nu, d(1) top, d(2) top, d(1) bottom, and d(2) bottom. The first three rows are pre-filled with data for materials 1, 2, and 3. Below the table is a large empty text area for additional input. At the bottom are "OK" and "Cancel" buttons.

Designation	E	Thickness	gamma	nu	d(1) top	d(2) top	d(1) bottom	d(2) bottom
	[kN/m²]	[m]	[kN/m²]	[-]	[m]	[m]	[m]	[m]
1	3.0000E+7	0.5137	0.0000	0.200	0.0500	0.0500	0.0500	0.0500
2	3.0000E+7	0.5412	0.0000	0.200	0.0450	0.0450	0.0450	0.0450
3	3.0000E+7	0.5824	0.0000	0.200	0.0450	0.0450	0.0450	0.0450

The numerical input for "**d(1)top**", "**d(2)top**", "**d(1)bottom**" and "**d(2)bottom**" are important for subsequent reinforcement of the slab. This defines the distance of the centroid of the reinforcement from the upper and lower slab surface for the principal directions of reinforcement **1** and **2**.

7.6.4 "Beam material" menu item

You can edit the parameters of any beams defined. A dialog box opens containing a number of input lines dependent on the number of beam materials specified in "**Boundary/Beams**" (Section 7.5.10).

Material	EJ [kN·m²]	GJ [kN·m²]
1	1.7222625E+7	1.0764140E+7
2	5.1200000E+5	3.2000000E+5

The following designations apply:

- EJ = Bending stiffness of beam
- GJ = Torsion stiffness of beam
- E = Young's modulus of beam
- G = Rigidity modulus of beam = $E/2 \cdot (1 + \nu)$
- J = Moment of inertia of beam

Due to the fractured tension zone often only a fraction of the theoretical torsional stiffness GJ (often even = 0.0) is adopted for reinforced concrete. If the values are not known they can be calculated by pressing the "**Calculate**" buttons.

7.6.5 "Test" menu item

The system is tested for the minimum node spacing and overlapping of triangle elements. The test can be aborted at any time by pressing the right mouse button.

7.6.6 "Analyse" menu item

7.6.6.1 Bandwidth optimisation/equation solver selection

After pointing to the "System/Analyse" or, alternatively, pressing [F5], the equation system bandwidth is examined and a message box displayed where necessary. If you do not agree with the bandwidth optimisation the analysis is cancelled.

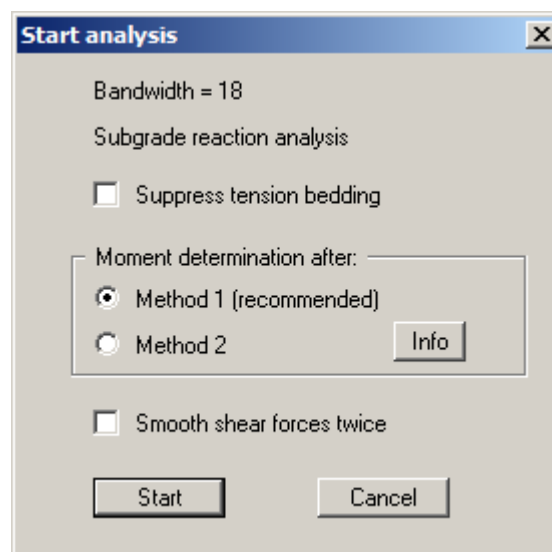
NOTE ON BANDWIDTH:

Finite-element methods give rise to equation systems which, depending on the system, only have values different to "0" in the region of the principal diagonals. The iterative equation solver contained in the FEM applications uses this in order to save memory and thus shorten computation time. The maximum distance between the principal diagonal and the system's *non-zero element most distant* from the principal diagonal is known as the bandwidth. In principle, iterative equation solvers do not require bandwidth optimisation. However, the results are much more stable and the computation time shorter if the bandwidth is small.

Following the bandwidth optimisation a dialog box opens where "Cholesky" or "Iterative" can be selected as equation solver. Information on the differences between the two equation solvers can be read by pressing the "Equation solver info" button. If you start the analysis by clicking "OK" different dialog boxes open depending on the selected analysis method (subgrade reaction modulus or constrained modulus).

7.6.6.2 Analysis using the subgrade reaction modulus method

The following analysis dialog box opens:



According to the theory of the elastically supported slab the subgrade reaction modulus also acts as a tensional spring, a physically impossible effect for slabs supported by the soil. If you activate the "Suppress tension bedding" the program checks for this effect after analysis and, if necessary, sets the subgrade reaction modulus for these areas to 0 and repeats the analysis. This is repeated until tension bedding is eliminated at all points. Please refer to Section 6.2 for details of the determination of moments.

7.6.6.3 Analysis using the constrained modulus method

The following analysis dialog box opens:

Start constrained mod. analysis

General

Max spacing settlement calculation [m]: 50.00

Displacement cancellation criteria [m]: 1.0000E-5

Damping (≥ 0.0 and < 0.95): 0.200

Max. allow. foundation press. [kN/m²]: 99999.0

Reduction ks for iter. max found. pr. [%]: 10.0

Limiting depth

☒ With limiting depth analysis Percentage: 20.0

Max. spacing for limiting depth [m]: 25.00

Determine limiting depth where? Slab centroid

Limiting depth info

Integrate settlements with:

☐ 7 points (Gauss) ☒ 13 points (Gauss)

☐ Analytic-numerical (precise but slow)

Moment determination after:

☒ Method 1 (recommended) ☐ Method 2 Info

☐ Smooth shear forces twice

Start Cancel

- **"General" group box**

The following condition must be adhered to when using the constrained modulus method:

Deflection curve of slab = settlement depression of elastic-isotropic half-space

In contrast to the subgrade reaction modulus method the constrained modulus method therefore requires an iteration process. In the first step the settlements (elastic-isotropic half-space) at all FEM mesh nodes are calculated for a constant load of 1 kN/m² on the FEM elements. The settlement resulting from the triangle loads at every FEM mesh node must be determined for this. For example, a mesh comprising 512 triangles and 289 nodes requires

$$512 \cdot 289 = 147,968 (!) \text{ settlement calculations.}$$

In an attempt to limit the settlement analysis time the **"Max. spacing settlement calculation"** allows input of a spacing. The settlements at system nodes are only determined from triangles with their centroids at a smaller distance to the system node than is defined here.

The end of iteration is defined using **"Cancellation criteria displacement"**. If the difference between the displacement of the slab and the calculated settlement (elastic-isotropic half-space) is smaller than the specified value at all points of the FEM mesh, the analysis is stopped.

The program recalculates the subgrade reaction moduli for the next iteration step from the quotients of the pressure at the node ($k_s \cdot w$) and the settlement (elastic-isotropic half-space) after every iteration step. In complicated systems this can lead to oscillations around the actual solution, the calculation does not converge. In this case, a **"Damping"** can be specified. The value **0.2** given in the dialog box has provided good results in the past. Values between 0.0 and 0.95 make the most sense. A value of 1.0 (complete damping) makes no sense and will therefore not be accepted by the program.

Theoretically infinitely high foundation pressures occur at the system boundaries when employing the constrained modulus method. This effect is reduced to stress peaks by use of finite analysis methods. If you need to avoid these stress peaks you may define a **"Max. allowable foundation pressure"**. After every iteration step the program checks whether this maximum foundation pressure has been exceeded. If it is exceeded the subgrade reaction modulus is reduced before carrying on with the next iteration step. This is done by calculating the subgrade reaction modulus at the appropriate points not from:

$$k_s \cdot w/s$$

but instead by using:

$$k_s = \text{max. allow. foundation pressure}/s$$

to calculate. Unfavourable numerical oscillations can also occur in the solution. You can therefore specify a maximum reduction of the subgrade reaction moduli at the appropriate locations so that a complete reduction is not immediately applied. Instead, the existing subgrade reaction modulus can be reduced by, for example, a maximum of 10% each time. Enter this percentage value in **"Reduction k_s for iter. Max. found. pr."**.

- **"Limiting depth"** group box

The limiting depth can be determined to DIN 4019. The **"Percentage"** defines the position of the limiting depth. The limiting depth is thus given by the condition:

$$\text{Soil surcharge} \cdot \text{percentage} = \text{stress below the triangle element}$$

The limiting depth can be calculated at given points of the slab (**"Slab centroid"**, **"Arbitrary point"**) or for every triangle element (**"Everywhere"**). Further information on the methods used can be read by pressing the **"Limiting depth info"** button.

- **"Integrate settlements with:"** group box

The settlement calculations for the elastic-isotropic half-space are performed using a Gaussian numerical integration. Integration using 7 points and 13 points is available. If you need higher precision (for slightly longer computation times) and are not superstitious, use integration with 13 points. The option of analytic-numerical determination of settlements is also provided. The analysis is precise, but takes a little longer.

- **"Moment determination after:"** group box

With regard to the determination of moments, please see the description in the section on the subgrade reaction modulus method (see Section 6.2).

A cancellation window opens after analysis starts. If you press the **"Cancel"** button the analysis is not immediately aborted. You may first edit all the parameters described above and, if required, continue with the analysis.

An analysis using the constrained modulus method takes considerably longer than the subgrade reaction modulus method due to the extensive settlement calculations necessary.

7.6.7 "Reinforce slab" menu item

Before concluding with the slab reinforcement, the system must be analysed. Once the analysis is complete you can select in the dialog box whether the reinforced concrete is designed using the old global safety factor concept to DIN 1045 or the new partial safety factor concept to EC 2/DIN 1045-1. The default method is always the safety factor concept selected in "File/New". If the safety factor concept is altered here it is also adopted for calculations. It may be necessary to reanalyse the system. Using the partial safety factor concept the following dialog box opens:

Reinforcement

Bemessung nach EC 2 / DIN 1045-1

Concrete C 30/37 Steel Str.st. 500/550

☒ Use Omega method (recommended)

Iteration precision: Medium

☒ Find minimum axial reinforcement

Partial safety factors

Permanent loads: 1.35

Changeable loads: 1.50

Reinforcement directions

Reinforcement direction (1) [°]: 0.0

Reinforcement direction (2) [°]: 90.0

(Reinforcement direction measured from the x-axis clockwise)

Shear design

☒ Perform shear design

Selected axial reinforcement [cm²]: 2.00

☐ Find minimum shear reinforcement
[for beam cross-sections only ($h/b \geq 0.25$)]

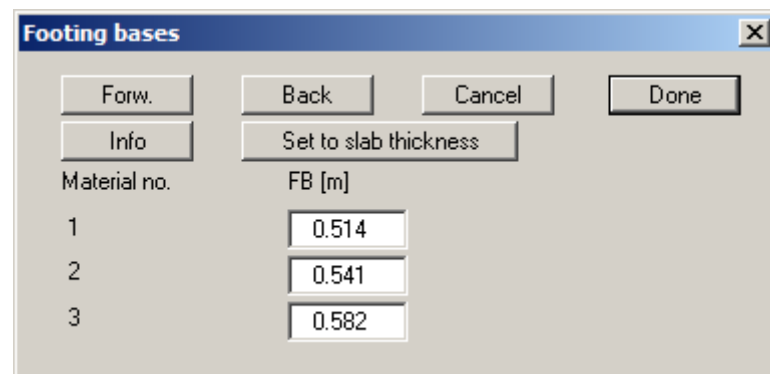
OK Cancel

The angle for the directions of reinforcement is measured positively clockwise from the horizontal axis.

The result is initially displayed in a message box, later in the reinforcement legend (see Section 7.7.11). This menu item can also be started using the **[F6]** function key.

7.6.8 "Footing bases" menu item

This menu item is only visible if you analyse using the constrained modulus method.



Material no.	FB [m]
1	0.514
2	0.541
3	0.582

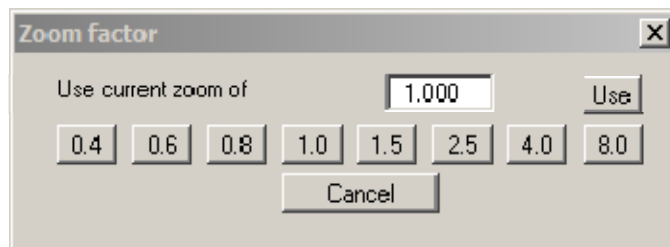
You can define footing bases for each material or accept the mean value using the "**Set to slab thickness**" button.

7.7 Graphics preferences menu

7.7.1 "Refresh and zoom" menu item

The program works on the principle of *What you see is what you get*. This means that the screen presentation represents, overall, what you will see on your printer. In the last consequence, this would mean that the screen presentation would have to be refreshed after every alteration you make. For reasons of efficiency and as this can take several seconds for complex screen contents, the screen is not refreshed after every alteration.

If, e.g., after using the zoom function (see below), only part of the image is visible, you can achieve a complete view using this menu item.



A zoom factor between 0.4 and 8.0 can be entered in the input box. By then clicking on "Use" to exit the box the current factor is accepted. By clicking on the "0.4", "0.6", etc. buttons, the selected factor is used directly and the dialog box closed.

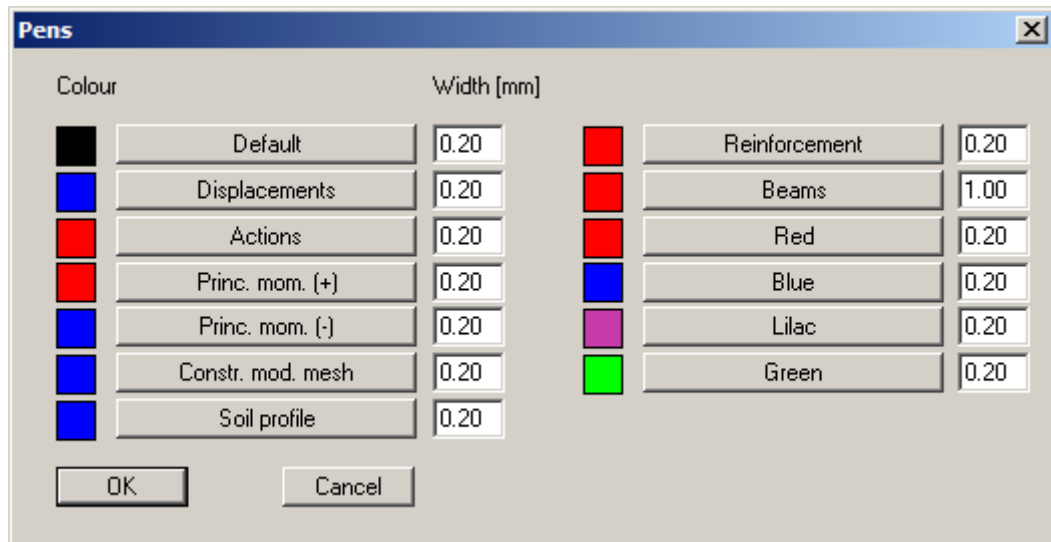
It is much simpler, however, to get a complete overview using [Esc]. Pressing [Esc] allows a complete screen presentation using the zoom factor specified in this menu item. The [F2] key allows screen refreshing without altering the coordinates and zoom factor.

7.7.2 "Zoom info" menu item

By clicking two diametrically opposed points you can enlarge a section of the screen in order to view details better. An information box provides information on activating the zoom function and on available options.

7.7.3 "Pen colour and width" menu item

In order to enhance the clarity of the graphics you can edit the pen settings for various graphic elements (e.g. moment, reinforcement, beams, etc.). You can edit the pen widths for the elements shown in the dialog box; by clicking on the button with the element designation you can also edit the pen or fill colours.



On *monochrome printers* (e.g. laser printers), colours are shown in a corresponding grey scale. Graphic elements employing very light colours may be difficult to see. In such cases it makes sense to edit the colour preferences.

7.7.4 "Legend font selection" menu item

With this menu item you can switch to a different true-type font. All available true-type fonts are displayed in the dialog box.

7.7.5 "Mini-CAD toolbar" and "Header toolbar" menu items

Using these two menu items you can add free text to the graphics and add lines, circles, polygons and images (e.g. files in formats BMP, JPG, PSP, TIF, etc.). The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the "**Mini-CAD**" manual provided. The differences between the "**Mini-CAD**" and "**Header CAD**" are as follows:

- Objects created with "**Mini-CAD**" are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location of any footings).
- Objects created with the "**Header CAD**" are based on the page format (in mm). This makes you independent of the coordinate system and keeps you in the same position on the page. You should select the "**Header toolbar**" if you wish to place general information on the drawing (company logo, report numbers, plan numbers, stamp etc.). Once you have saved the header information to disk (see "**Mini-CAD**" user manual), you can load it into completely different systems (with different system coordinates). The saved header information will appear in exactly the same position on the page, which greatly simplifies the creation of general page information.

7.7.6 "Toolbar preferences" menu item

After starting the program a horizontal toolbar appears below the program menu bar. If you would rather work with a popup window with several columns, you can specify your preferences using this menu item. The smart icons can also be switched off.

At the bottom of the program window you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "GGU-SLAB.alg" file (see menu item "Graphics preferences/Save graphics preferences") and will be active at the next time the program is started.

By clicking on the tools (smart icons) you can directly reach most of the program functions. The meaning of the Smart icons appears as a text box if you hover with the mouse pointer over the tools. Some of the tool functions can be activated from the normal menu items.



"Next page"/"Previous page"

Using this icon, you can navigate between the individual pages in the *tabular representation*.



"Select page"

If you are in the *tabular representation*, you can use this icon to jump to a specific page or to return to the *normal representation*, that is, to the graphics.



"Zoom out"

If you have previously *zoomed in*, this tool returns to a full screen display.



"Zoom (-)"/"Zoom (+)"

With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.



"Colour on/off"

If you need to remove the colour from the system presentation, to create a black and white print-out, for example, use this on/off switch.



"Copy/print area"

Use this tool to copy only parts of the graphics in order to paste them, e.g. to a report. You will see information on this function and can then mark an area, which is copied to the clipboard or can be saved in a file. Alternatively you can send the marked area directly to your printer (see "Tips and tricks", Section 8.3).



"Undo"

By clicking this tool the last performed input or change or movement of graphical elements (e.g. made using [F11]) can be undone.




"Restore"

By clicking this tool, the last change undo carried out using "Undo" can be restored.

7.7.7 "3D toolbar" menu item

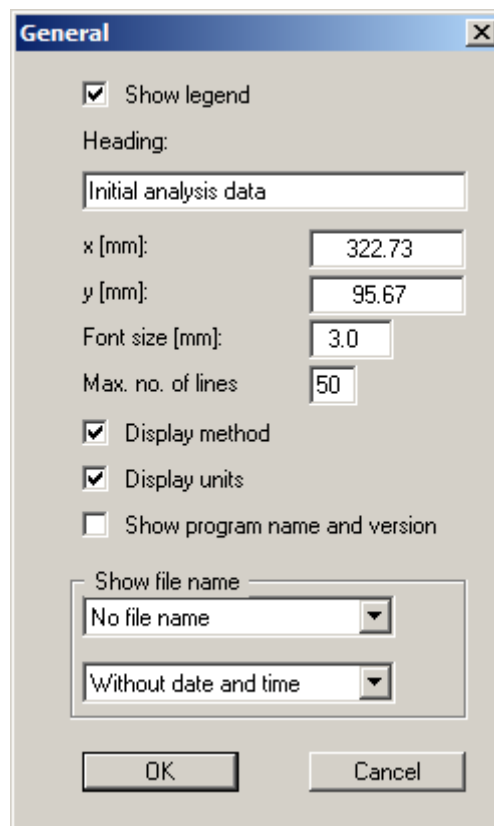
This menu item allows you to choose a pop-up window with tools for rotating and zooming three dimensional images. If you hover with the mouse pointer over the tools, the meaning of the Smart icons appears in a text box.

The tools in this pop-up 3D window allow you to rotate the graphics around one of the three axes. The plus and minus signs designate the direction of rotation. The angle of rotation (default: 45°)

can be adjusted as wished using the  tool.

7.7.8 "General legend" menu item

A legend with general properties will be displayed on your output sheet if you have activated the "Show legend" check box. Using this menu item you can alter the type of presentation.



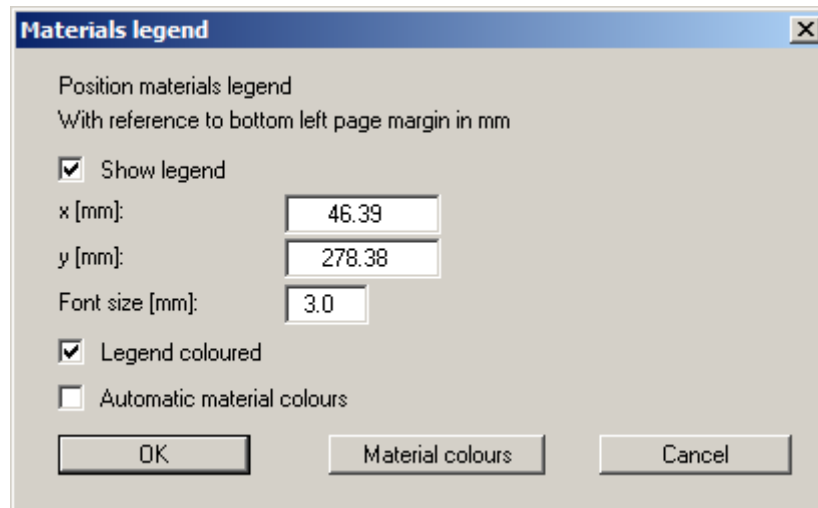
Beside the heading input the display of the used method and units can be activated. Any project identification entered (see Section 7.6.2) will also be shown in the general legend.

You can define and edit the position of the legend using the values "x" and "y". You control the size of the legend using "Font size" and "Max. no. of lines"; where necessary, several columns are used. The fastest way to modify the position of the legend is to press the [F11] function key and then to pull the legend to the new position with the left mouse button pressed.

In the general legend you can, if wished, display information on the program and on the current file. The current file name can be shown in the legend, either with or without the path. Date and/or time can be displayed, too.

7.7.9 "Materials legend" menu item

A legend with the slab materials will be displayed on your output sheet if you have clicked the "**Show legend**" check box. Using this menu item you can alter the type of presentation or turn off the legend completely.



You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**".

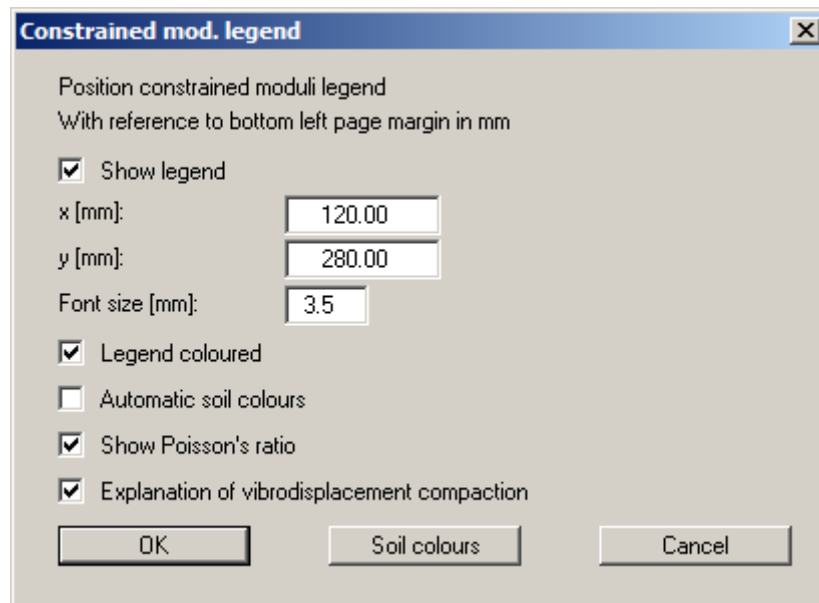
The fastest way to modify the position of the legend is to press the [F11] function key and then to pull the legend to the new position with the left mouse button pressed.

- "**Legend coloured**"
The material colours are displayed in the legend. Otherwise, they will be numbered.
- "**Automatic material colours**"
The materials are assigned colours automatically by the program. If the check box is not selected, the colours individually defined using the "**Material colours**" button will be adopted.
- "**Material colours**"
You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each soil layer a new number or reorganise using the "**Material colours/Reorganise**" command button. You can save your colour preferences to a file with "**Material colours/Save**" and use them for different systems by means of the "**Material colours/Load**" command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the "**Info**" button.

The material colours specified using this menu item are - regardless of whether activated for the legend - always used for visualisation of the FEM mesh, assuming the "**Coloured**" check box was activated in the preferences (see menu item "**FEM mesh/Preferences**", Section 7.2.1).

7.7.10 "Constrained mod. legend" menu item

When using the constrained modulus method you can display a legend comprising the constrained modulus soil properties by activating the "**Show legend**" check box.



You can define and edit the **position** of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**". The fastest way to modify the position of the legend is to press the [F11] function key and then to pull the legend to the new position with the left mouse button pressed.

- "**Legend coloured**"
The soil colours are displayed in the legend. Otherwise, they will be numbered.
- "**Automatic soil colours**"
The soils are assigned soils colours automatically by the program. If the check box is not selected, the soil colours individually defined using the "**Soil colours**" button will be adopted.
- "**Show Poisson's ratio**"
Poisson's ratio can be switched off in the legend, for example if Poisson's ratio $\nu = 0.0$ for all soils (see Section 7.4.3).
- "**Explanation of vibrodisplacement compaction**"
The explanation of vibrodisplacement compaction can be displayed in legend by activating this check box.
- "**Soil colours**"
You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each soil layer a new number or reorganise using the "**Soil colours/Reorganise**" command button. You can save your colour preferences to a file with "**Soil colours/Save**" and use them for different systems by means of the "**Soil colours/Load**" command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the "**Info**" button.

The soil colours specified using this menu item are - regardless of whether activated for the legend - always used for visualisation of the constrained moduli, assuming the "**Coloured constrained mod.**" check box was activated in the preferences (see menu item "**Constrained mod. / Preferences**", Section 7.4.2).

7.7.11 "Reinforcement legend" menu item

If the reinforcement has been visualised using the '**Evaluation**' menu, a legend containing general reinforcement data can be displayed on the screen. Activate the "**Show legend**" check box in this menu item's dialog box.

The position of the legend is defined by the values "**x**" and "**y**". The size of the legend is controlled using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used.

The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

7.7.12 "Beam legend" menu item

If you activate the "**Show legend**" check box in the dialog box for this menu item, a legend containing general information on any beams is displayed on the screen. The position of the legend is defined by the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**".

The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

7.7.13 "Section legend" menu item

If you activate the "**Show section course**" check box in the dialog box for this menu item, a legend containing a sketch showing the location of the section is displayed on the screen. The position of the legend can be defined and edited using the values "**x**" and "**y**", "**Width**" and "**Height**". The legend heading and font size can be modified.

7.7.14 "Move legends" menu item

Select this menu item in order to position legends at the desired position on the output sheet. You can also move legends by pressing [**F11**] and then positioning the legend box with the left mouse button pressed. In that case an info-box appears no more.

7.7.15 "Save graphics preferences" menu item

Some of the preferences you made with the menu items of the "**Graphics preferences**" menu can be saved to a file. If you select "**GGU-SLAB.alg**" as file name, and save the file on the same level as the program, the data will be automatically loaded the next time the program is started and need not be entered again.

If you do not go to "**File/New**" upon starting the program, but open a previously saved file instead, the preferences used at the time of saving are shown. If subsequent changes in the general preferences are to be used for existing files, these preferences must be imported using the menu item "**Graphics preferences/Load graphics preferences**".

7.7.16 "Load graphics preferences" menu item

You can reload a graphics preferences file into the program, which was saved using the "**Graphics preferences/Save graphics preferences**" menu item. Only the corresponding data will be refreshed.

7.8 Page size + margins menu

7.8.1 "Auto-resize" menu item

This menu item provides a to-scale visualisation, in both x and y coordinates, of the system and result graphics. If you have previously altered the image coordinates using the mouse or via editor, you can quickly achieve a complete view using this menu item. This function can also be accessed using the [F9] function key.

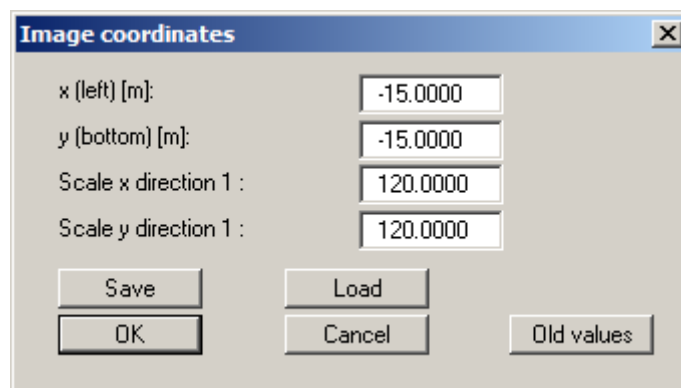
7.8.2 "Manual resize (mouse)" menu item

You can use the coordinates of a section of the visualisation as the new image coordinates by marking the desired area with the mouse, pressing the left mouse button and holding the [Ctrl] and [Shift] keys. The scales of the x- and y-axes are adjusted accordingly. If the previous proportions (scale x-direction/scale y-direction) need to be retained, the "**Proportional section**" check box must be activated.

Alternatively, you can simply "**Redefine origin**" of the visualisation. The previous scale preferences are not affected by this.

7.8.3 "Manual resize (editor)" menu item

You can alter the image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the *drawing area*. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 7.8.4).



The image coordinates entered here can be saved in a file with the extension ".bxy" and be re-loaded later for the same file or for different files.

If you want to recover the previous values during input or use the menu item again after editing the coordinates, you can do this by pressing the "**Old values**" button.

7.8.4 "Page size and margins" menu item

The default page set-up is A3 when the program is started. You can edit the page format in the following dialog box.

The screenshot shows a 'Page size' dialog box with the following fields and values:

Section	Field	Value
Page in general	Height =	297.00
	Width =	420.00
Page margin in mm	Left =	25.00
	Right =	8.00
	Top =	8.00
	Bottom =	8.00
Plot margin in mm	Left =	20.00
	Right =	20.00
	Top =	40.00
	Bottom =	15.00

- **"Page in general"** defines the size of the output sheet. The A3 format is set as default. The program automatically draws thin cutting borders around the page, which are required when using a plotter on paper rolls. The borders can be switched off using the **"With borders"** check box in the **"Page size + margins/Margins + borders"** menu item (see Section 7.8.6).
- **"Page margin"** defines the position of a frame as a distance to the margins. This frame encloses the subsequent diagram. You can switch off the frame deactivating the **"With margins"** check box in the **"Page size + margins/Margins + borders"** menu item (see Section 7.8.6).
- The **"Plot margin"** define a set distance between the page margin and the actual *drawing area* in which the graphical evaluation of your input is presented.

7.8.5 "Font size selection" menu item

You can edit font sizes for labelling the various drawing elements.

The font sizes of text within legends are edited in the respective legend editor. Just double-click in a legend to do this.

7.8.6 "Margins + borders" menu item

The program automatically draws thin cutting edges around the page, which are required when using a plotter on paper rolls. Page margins (see menu item "**Page size + margins/Page size and margins**") defines the position of a frame as a distance to the cutting border. This frame encloses the subsequent diagram. You can switch off the lines by deactivating the "**With margins**" and "**With borders**" check boxes.

7.8.7 "Draw Mini-CAD first" menu item

Objects added using **Mini-CAD** are normally drawn via the system representation. If the **Mini-CAD** objects are to be positioned behind the graphics this can be specified in the pop-up menu of the **Mini-CAD** module (layer dialog in **Mini-CAD/"Presentation"** button).

7.8.8 "Undo" menu item

If you have carried out any change in your system (e.g. moving nodes, assigning boundary conditions) or moved objects to a different position on the screen after selecting the "**Graphics preferences/Move objects**" menu item or using the **[F11]** function key, this menu item will allow you to undo the changes. This function can also be reached by using the key combination **[Alt] + [Back]** or the appropriate tool in the toolbar (see Section 7.7.6).

7.8.9 "Restore" menu item

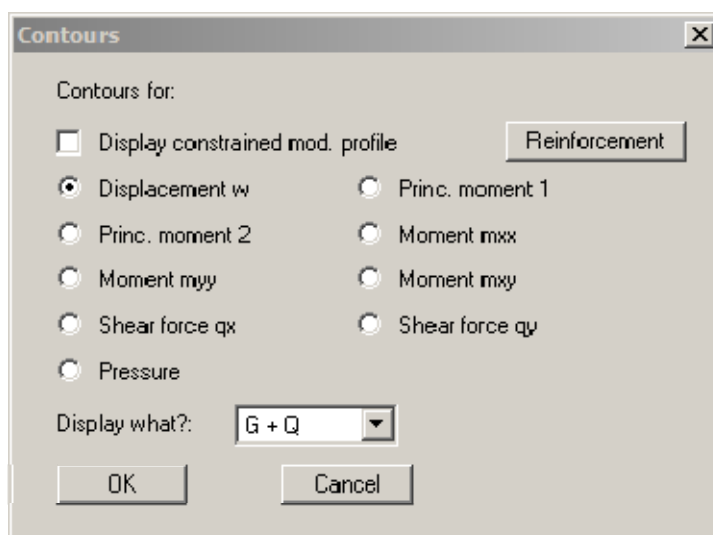
When this menu item is selected the last change made in your system or the last change in the position of objects, which you undid using the "**Page size + margins/Undo**" menu item will be restored. This function can also be reached by using the key combination **[Ctrl] + [Back]** or the appropriate tool in the toolbar (see Section 7.7.6).

7.8.10 "Preferences" menu item

You can activate or deactivate the undo functions.

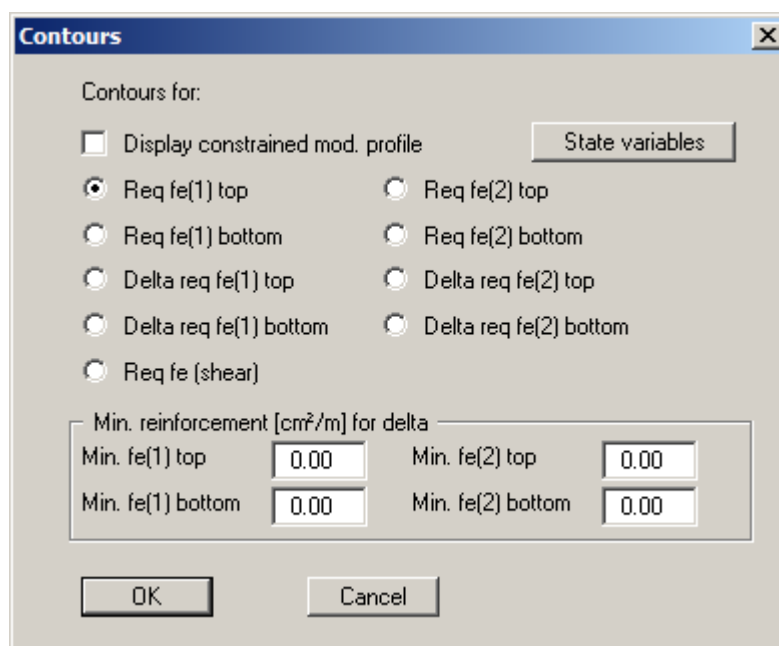
7.9.1 General information on representation of contour lines, circles and tables

Numerous options are available for evaluation and graphical representation. If you point to one of the contour line diagrams or the menu item "**Evaluation/Circles**" or "**Evaluation/Table**", you must first select the parameter to be displayed. The following box opens if you are working with the *partial factor concept*.



In addition to the required state variable, decide whether the display should include permanent (G) and/or live loads (Q).

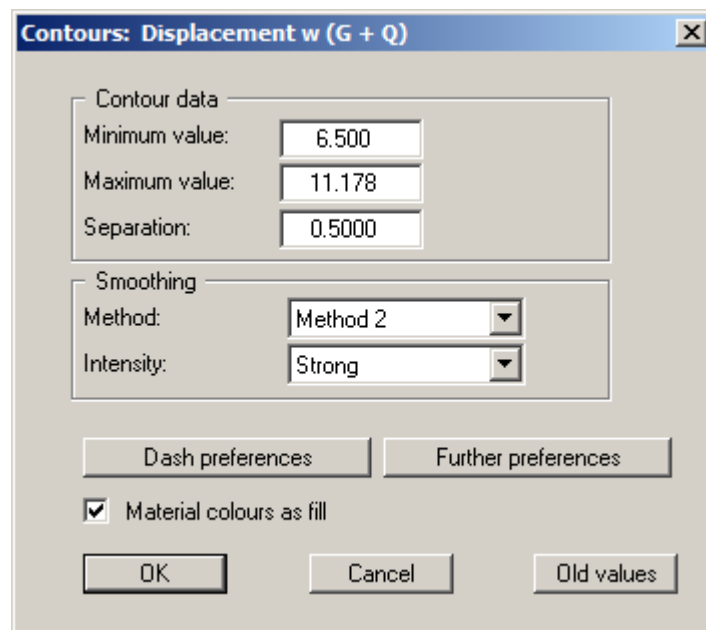
Alternatively, the reinforcement of reinforced slabs can also be visualised. Click the "**Reinforcement**" button and the following dialog box opens:



Using the four upper option buttons you select the required reinforcement at the upper and lower slab faces and for the two reinforcement directions, **1** and **2**. If the slab has a minimum reinforcement you can enter the minimum reinforcement for the upper and lower slab faces (reinforcement directions **1** and **2**) in "**Min. reinforcement [cm²/m] for delta**" and then have the further reinforcement requirement displayed using, for example, the "**delta req fe(1) top**" option button. If you press the "**State variables**" button the dialog box for selecting the state variables opens again (see above).

7.9.2 "Normal contours" menu item

After selecting the required parameter you will see the following dialog box where you can specify preferences for the display of contour lines.



The program shows you the existing smallest and largest values and the spacing of the contour lines. If you want the visualisation to begin at a different value, the initial value can be entered here. You can also vary the spacing, for example, in order to reduce the number of contour lines drawn.

When you select this menu item the settings displayed here are always those automatically selected by the program. Using the "**Old values**" button, the preferences used for the previous contour line diagram are adopted. This information is saved with the record.

You can also select from three smoothing out procedures:

- "**Do not smooth**"
As a linear method is implemented within the triangle elements, this will provide analysis results without smoothing performed by the program.
- "**Method 1**" and "**Method 2**"
these smoothing out procedures employ two different Bezier splines. Smoothing out intensities can be given for both in order to acquire smooth contour lines. **Method 2** creates very "**round**" contours, with the danger that the true results may be falsified.

The "**Dash preferences**" and "**Further preferences**" buttons can be used to specify the type of visualisation of the contour lines.

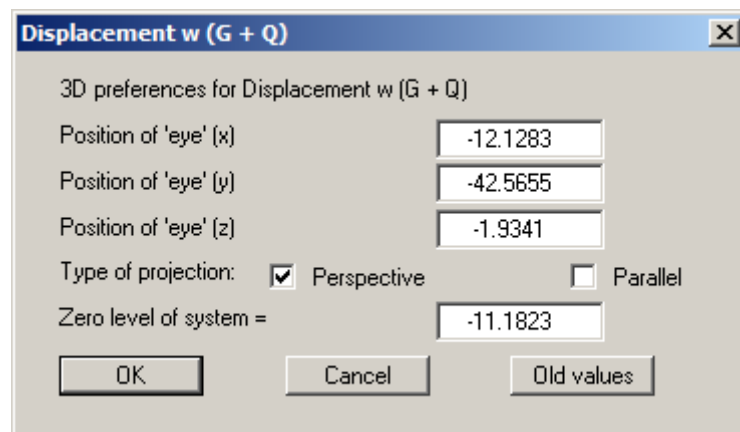
7.9.3 "Coloured" menu item

After selecting the required parameter the lines of equal state variables or the reinforcement parameters can be displayed in colour.

The preference options for the dialog box were explained in the "**Constrained mod./Layer base contours**" menu item (see Section 7.4.5).

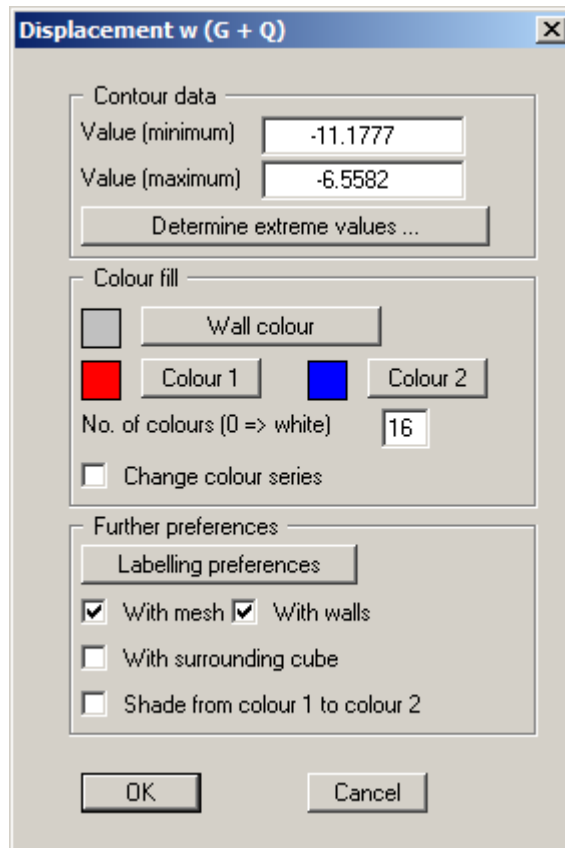
7.9.4 "3D" menu item

After selecting the required parameter the lines of equal state variables or the reinforcement parameters can be displayed in 3D for the existing nodes. In the following dialog box you can choose between perspective and parallel projection, as well as selecting the position of the "**eye**".



The program will generally make sensible suggestions for the input values in the dialog box, so you will not normally need to make alterations.

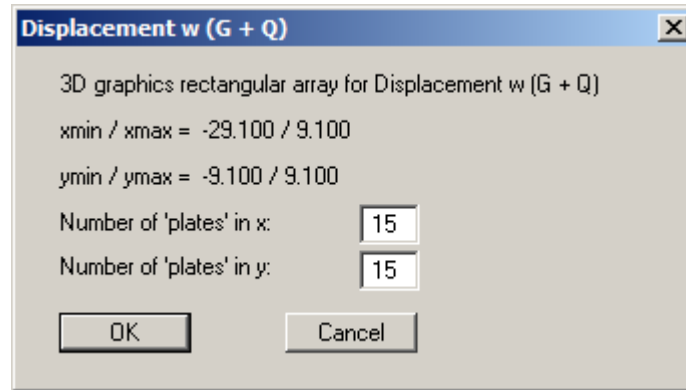
After leaving the dialog box by pressing "**OK**" you will see the following dialog box. It greatly resembles the box shown in the menu item "**Constrained mod./Layer base contours**".



You can also specify whether an enclosing cube should be displayed; in some cases this improves the 3D effect. The "**Shade from colour 1 to colour 2**" check box represents a bit of a special effect. If this check box is activated a light source is simulated in the region of the "**eye**". The angle between the light beam and the respective 3D surface represents a measure of the reflection. The area is shaded in accordance with colours 1 and 2. A good choice of colours, for example, would be dark grey for colour 1 and pale grey for colour 2.

7.9.5 "3D array" menu item

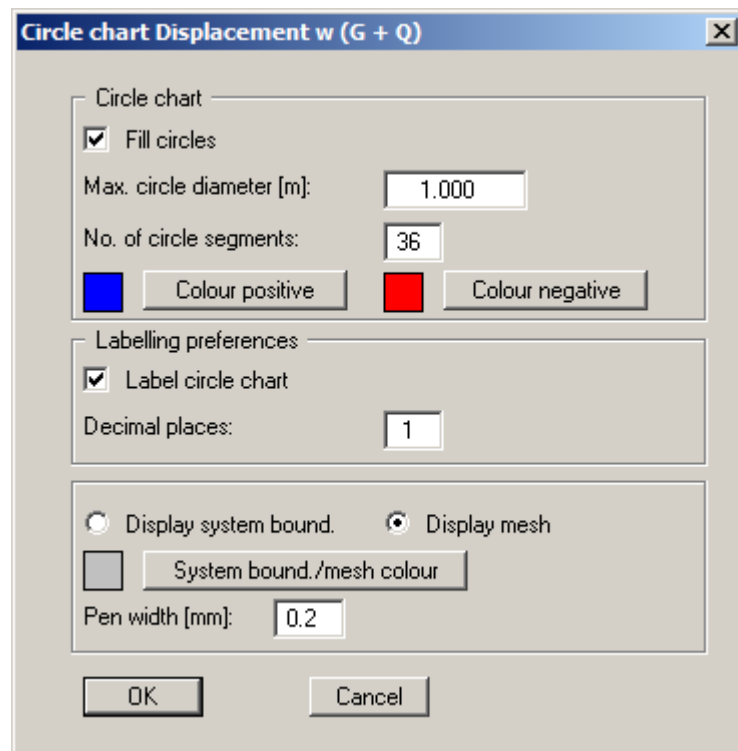
The 3D effect is usually lost in an irregular triangle mesh. The "**Evaluation/3D array**" menu item is useful here. After selecting the required parameter the following dialog box opens where you can define the array.



After leaving the dialog box by pressing "**OK**" you will at first see the same box as that described in "**Evaluation/3D**". The program calculates the state variables at the array points by linear interpolation of the results of the FEM analysis. You then see the dialog box for coloured contours preferences as described in "**Evaluation/3D**".

7.9.6 "Circles" menu item

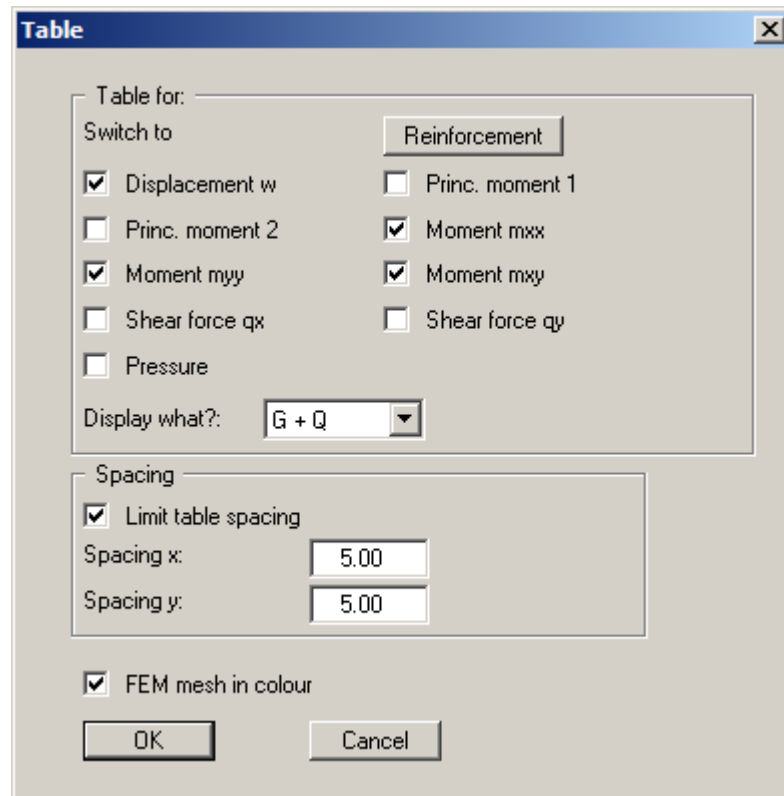
After selecting the required parameter a circle chart can be generated containing the appropriate state variable as differently sized (value-dependent) circles. Using result visualisation in circle format it is possible to swap between state variables and reinforcement in analogy to the contour line visualisation dialog box (see dialog box in Section 7.9.1).



If you activate the "**Fill circles**" check box the state variables are represented with a colour fill defined using the "**Positive colour**" or "**Negative colour**" buttons, corresponding to their sign. The "**Max. circle diameter**" defines the circles diameter for the largest state variable. The "**Number of circle segments**" specifies the resolution of the circle. For example, triangles are represented for a value of **3**. Each circle can be annotated with the value of the state variable. The system boundaries or the FEM mesh can be drawn in addition to the circles. The colour and width of the pens can be specified for the graphical visualisation.

7.9.7 "Table" menu item

All state variables can be displayed graphically in a table. Using result visualisation in table format it is possible to swap between state variables and reinforcement in analogy to the contour line visualisation dialog box (see dialog box in Section 7.9.1).

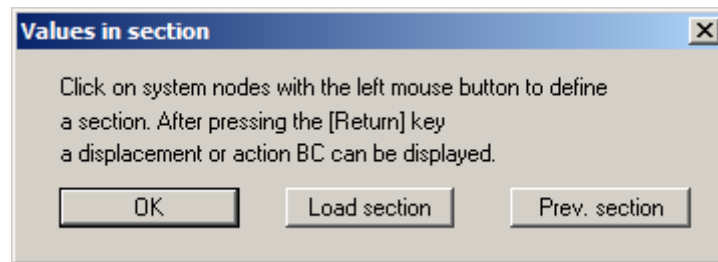


A number of state variables can be selected in the dialog box. They are represented in the graphics directly after closing the box using "OK". The font size used in the tables can be specified using the menu item "**Page size + margins/Font size selection**" (see Section 7.8.5).

7.9.8 "Values in node section" menu item

You can display state variables in a section. To do this, define a section by clicking the appropriate nodes using the left mouse button. Intermediate points are inserted automatically by the program. The last point in the list is deleted by clicking the right mouse button. When the section is located where you want it press **[Return]**. Select the state variable to be represented.

You will first see a dialog box for specifying preferences for the section representation, such as labelling or representation as a *fence diagram* at FEM. Here, you also have the option of saving the section to a file using the **"Save section"** button. If you click this menu item a second time the following dialog box opens:



By pressing **"OK"** you can now create a new section or, using the other buttons, either return to the previous section or load one saved earlier.

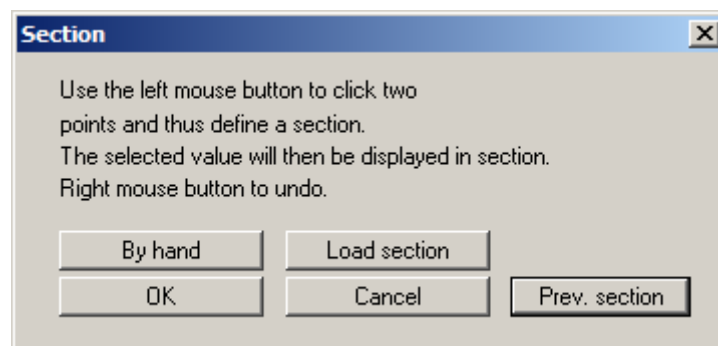
If you have activated **"Show section course"** using the menu item **"Graphics preferences/Section legend"** (see Section 7.7.13), a location sketch is automatically displayed on the screen showing the slab outline and the defined section.

7.9.9 "Position of node section" menu item

You can display the location of the section defined using the menu item **"Evaluation/Values in node section"** and send it to the printer.

7.9.10 "Any section" menu item

After selecting the required parameter you will see the following dialog box for defining the section:



For this type of evaluation the section course is no longer bound to the FEM nodes, but consists of a start and an end point. Using the **"By hand"** button you can specify the start and end points using numerical values instead of the mouse. If you have previously saved a section it can be loaded again using **"Load section"**. The **"Prev. section"** button is only visible if a section has already been defined and loads the section defined immediately prior.

After completing the section definition by pressing **[Return]** the number of subdivisions can be specified in a further dialog box. This defines the number of points are required for calculation of the sectional values. After confirming with **"OK"** the following dialog box opens:

You can now specify whether the section should be displayed in colour. You can also define the intercept of the x-axis with the y-axis by entering a value for **"y0"**. In this case the **"Use"** check box must be activated. If you do not use this option, the x axis will intercept the y axis at the smallest y value. The section can also be displayed as a *fence diagram*.

After clicking the "**Calculate torsional stiffness**" button, enter the moment and the start and end points of the section. The input data and the results of the torsional stiffness calculations are subsequently displayed in a message box. The data then be copied to the Windows clipboard by pressing the "**Copy data to clipboard**" button. After confirming the message box you are returned to the original section editor box. The calculated torsional stiffness is later displayed in the *General legend*.

If you need to display the soil layering, activate the "**With layering**" check box. Using the "**Factor for values**" you can alter the size of the displayed state variable or the reinforcement. The location of the zero point for visualisation of the state variables within the layering visualisation is defined between 0 (top) and 1 (bottom). In addition, pressure and footing bases can be displayed.

You can save the section in order to load it again at a later date (see above).

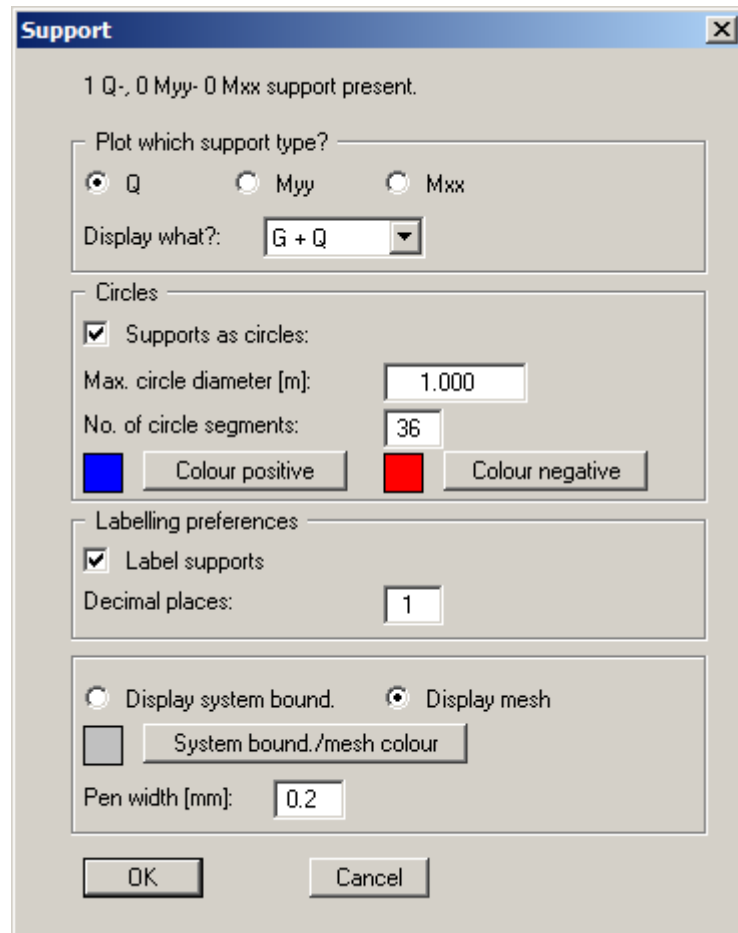
If you have activated "**Show section course**" using the menu item "**Graphics preferences/Section legend**" (see Section 7.7.13), a location sketch is automatically displayed on the screen showing the slab outline and the defined section.

7.9.11 "Position of any section" menu item

You can display the location of the section defined using the menu item "**Evaluation/Any section**" and send it to the printer.

7.9.12 "Support" menu item

You can generate a circle chart of the support forces using this menu item. The current supports are shown at the top of the dialog box.



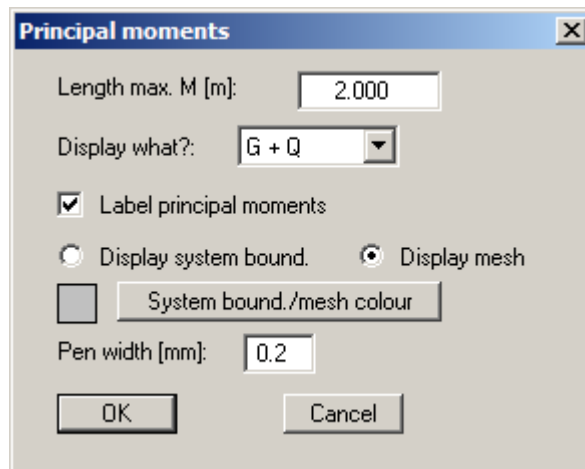
Select the support to be displayed in the upper group box of the dialog box.

- Q = support resulting from displacement boundary condition w
- Mxx = support resulting from displacement boundary condition $\phi(y)$
- Myy = support resulting from displacement boundary condition $\phi(x)$

All other input is identical to the input in menu item **"Evaluation/Circles"** (see Section 7.9.6).

7.9.13 "Principal moments" menu item

The principal moments can be displayed as value-dependent lines. The following dialog box opens if you have selected the *partial safety factor concept*.



The line colour and line thickness can be specified separately for positive and negative principal moments in the menu item "**Graphics preferences/Pen colour and width**" (see Section 7.7.3). The lines for positive principal moments have a small vertical line at both ends.

7.9.14 "Reinforcement" menu item

The reinforcement is displayed as lines according to direction and size. Shear reinforcement can also be displayed as a circle diagram; otherwise the numerical values are displayed.

Reinforcement

Which reinforcement?

☐ Top ☐ Bottom ☒ Shear

Length max. fe [m]: 0.000

Shear reinforcement circle chart

☒ Circular shear reinforcement

Max. circle diameter [m]: 0.500

No. of circle segments: 36

☒ Label reinforcement

☐ Display system bound. ☒ Display mesh

System bound./mesh colour

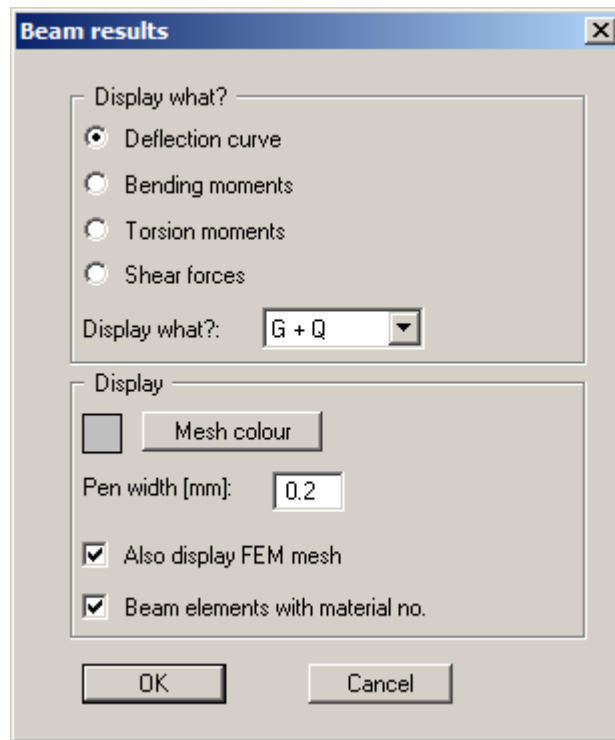
Pen width [mm]: 0.2

OK Cancel

The line colour and line thickness, and the labelling colour can be specified in the menu item **"Graphics preferences/Pen colour and width", "Reinforcement"** check box (see Section 7.7.3).

7.9.15 "Beams" menu item

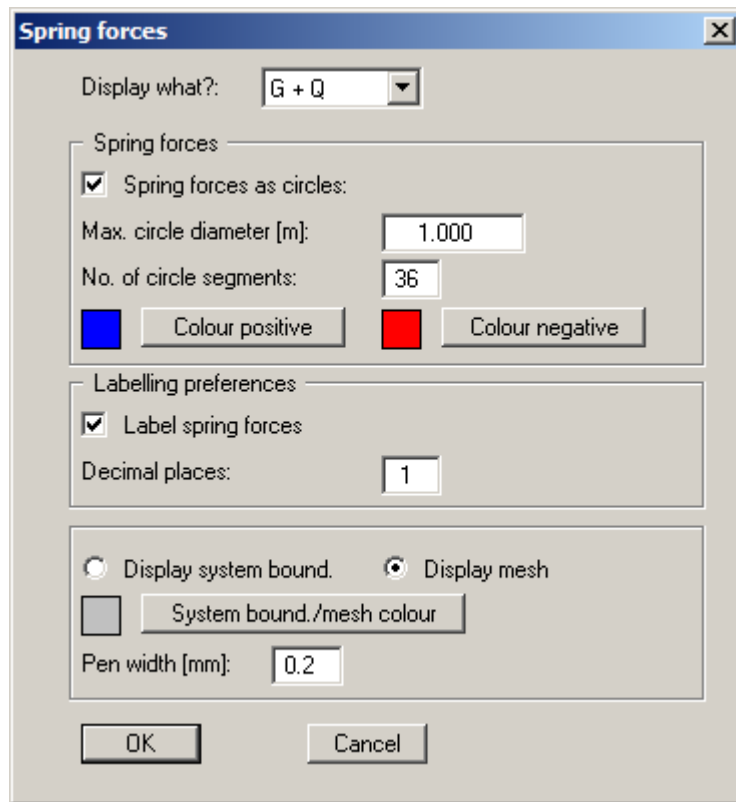
If there are beams in the system the state variables for these beams can be displayed.



Depending on the selected state variable you will see a further dialog box displaying the maximum values. You can then specify further preferences for the display, such as labelling.

7.9.16 "Single springs" menu item

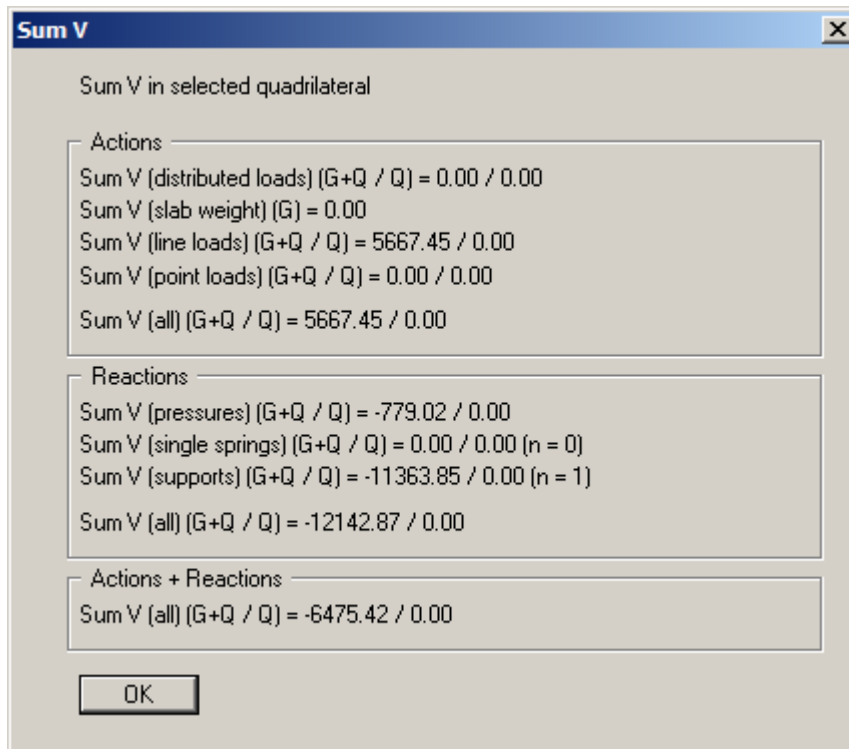
If there are single springs in the system they can be displayed as a circle chart.



The dialog box is almost identical to the dialog box in menu item "**Evaluation/Circles**" (see Section 7.9.6).

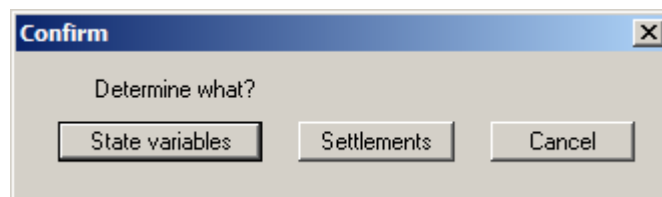
7.9.17 "Sum V" menu item

If supports, pressures or single springs are available, sum V can be determined for any areas. You must click on four points in an anti-clockwise direction. The results will then be displayed in a message box.



7.9.18 "Individual values" menu item

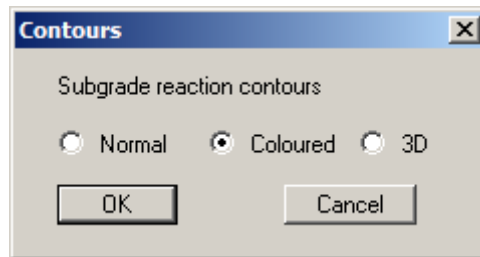
By clicking a node using the left mouse button all data for this node can be viewed. If you are analysing a problem using the constrained modulus method, you may first decide whether to determine the state variables or the settlements.



If you click the "**Settlements**" button the program carries out a settlement calculation for every clicked point for the elastic-isotropic half-space. These points are then no longer connected to the system nodes. They may be located outside of the slab as long as a constrained modulus profile is defined at this particular point.

7.9.19 "Subgrade reactions" menu item

When using the subgrade reaction modulus method you can draw the contour lines of the subgrade reaction moduli resulting from the program's iteration analysis.



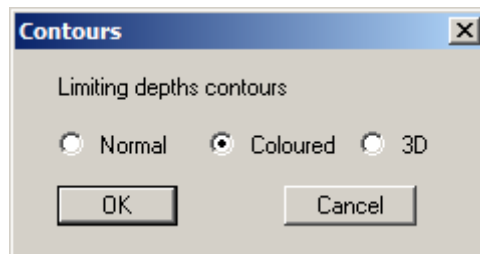
Define the type of contour diagram using the "**Normal**", "**Coloured**" and "**3D**" option buttons. Descriptions of the various types of contour lines can be found in Sections 7.9.2 to 7.9.4.

7.9.20 "Difference" menu item

When using the constrained modulus method you can display the differences between the settlements s (elastic-isotropic half-space) and the displacements w (bedded slab) as contour lines. Select the type of contour in the dialog box by means of the "**Normal**", "**Coloured**" and "**3D**" option buttons. Descriptions of the various types of contour lines can be found in Sections 7.9.2 to 7.9.4.

7.9.21 "Limiting depths" menu item

If you have chosen to calculate the limiting depth for every triangle element when analysing your system ("**Everywhere**", see Section 7.6.6.3), the calculated limiting depths can be visualised as contour lines using this menu item.

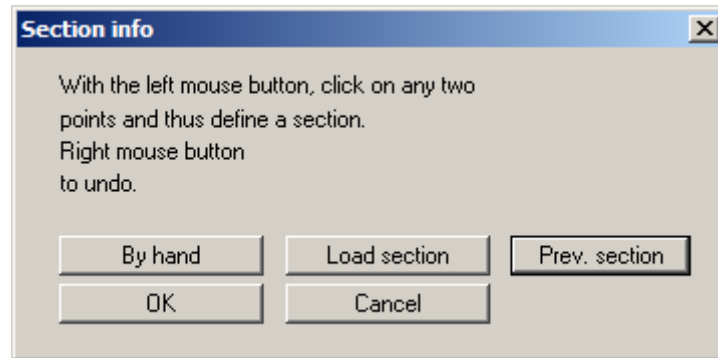


Define the type of contour diagram using the "**Normal**", "**Coloured**" and "**3D**" option buttons. Descriptions of the various types of contour lines can be found in Sections 7.9.2 to 7.9.4.

7.10 Special menu (constrained modulus method only)

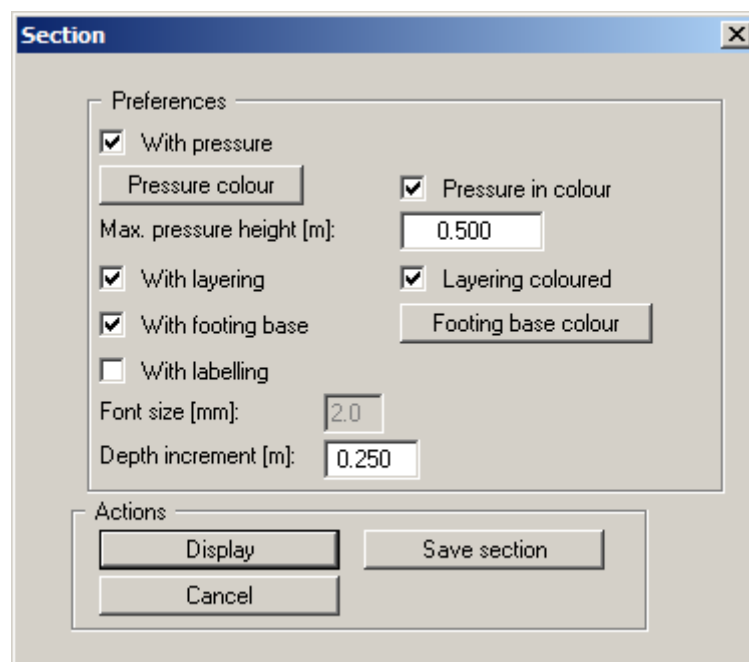
7.10.1 "Define stress section" menu item

The stresses for any given section can be displayed using this menu item. Specify the means of defining the section in the dialog box.



For this type of evaluation the section course is no longer bound to the FEM nodes, but consists of a start and an end point. Using the **"By hand"** button you can specify the start and end points using numerical values instead of the mouse. If you have previously saved a section it can be loaded again using **"Load section"**. The **"Prev. section"** button is only visible if a section has already been defined and loads the section defined immediately prior.

After completing the section definition by pressing **[Return]** the number of subdivisions can be specified in a further dialog box. This defines the number of points are required for calculation of the sectional values. After confirming with **"OK"** the following dialog box opens:



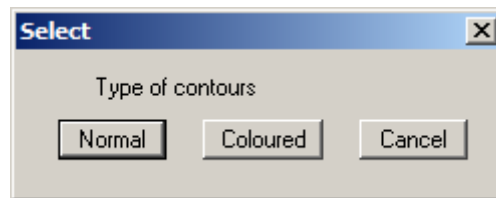
You can specify a number of preferences for displaying the stresses. After leaving the dialog box via the **"Display"** button the program first calculates the stresses and then displays the stress distribution at depth for the specified points. It is also possible to save the section from within the dialog box.

7.10.2 "Preferences" menu item

If you have already defined a section and displayed it on the screen, you may retroactively edit the preferences for the graphical representation using this menu item. A dialog box for the section representation opens; this can also be reached by going to the menu item **"Special/Define stress section"** (see Section 7.10.1). After leaving the dialog box the stresses are first determined once again and then the section displayed.

7.10.3 "Stress bulb section" menu item

When you initially go to this menu item the same dialog boxes open as shown in **"Special/Define stress section"** in Section 7.10.1. However, after leaving the graphical preferences dialog box by using the **"Display"** button you select the type of contours.



Once you have selected the type of contours required the program calculates the stresses along the specified section. You then see the familiar dialog boxes for the normal or the coloured contour line diagram (see Sections 7.9.2 and 7.4.5).

7.10.4 "(Stress bulb section) Normal contours" menu item

If the stresses for the specified section have already been calculated using the previous menu item **"Special/Stress bulb section"** you can move directly to the normal contour diagram using this menu item. The familiar dialog box opens (see Section 7.9.2).

7.10.5 "(Stress bulb section) Coloured" menu item

If the stresses for the specified section have already been calculated using the menu item **"Special/Stress bulb section"** you can move directly to the coloured contour diagram using this menu item. The familiar dialog box opens (see Section 7.4.5).

7.10.6 "Vertical settlements section" menu item

Specification of the section for displaying the vertical settlements is in analogy to the menu item **"Special/Define stress section"** (see Section 7.10.1). If you close the dialog box for the graphical representation preferences using the **"Display"** button, the program first calculates the settlements in the user-defined section. You will then see a combo box for selecting the type of contour representation.

Once you have selected the type of contours required the program calculates the settlements along the specified section. You then see the familiar dialog boxes for the normal or the coloured contour line diagram (see Sections 7.9.2 and 7.4.5).

7.10.7 "(Vertical settlements section) Normal contours" menu item

If the vertical settlements for the specified section have already been calculated using the previous menu item **"Special/Vertical settlements section"** you can move directly to the normal contour diagram using this menu item. The familiar dialog box opens (see Section 7.9.2).

7.10.8 "(Vertical settlements section) Coloured" menu item

If the vertical settlements for the specified section have already been calculated using the menu item **"Special/Vertical settlements section"** you can move directly to the coloured contour diagram using this menu item. The familiar dialog box opens (see Section 7.4.5).

7.10.9 "Settlement depression" menu item

This menu item allows the calculation and visualisation of settlement depressions. You will see almost exactly the same dialog box for defining the section as described in **"Evaluation/Any section"** (see Section 7.9.10). For this type of evaluation the section course is not bound to the FEM nodes either, but consists of a start and an end point

After completing the section definition by pressing **[Return]**, you define the number of points at which section values are calculated by entering the number of subdivisions into a dialog box. You will see the dialog box previously described in the menu item **"Evaluation/Any section"**, in which you can define preferences for settlement depression visualisation.

7.10.10 "Stresses for GGU-CONSOLIDATE" menu item

If a stress section was previously defined (see Section 7.10.1), the stress distributions at the analysed points can be exported to the **GGU-CONSOLIDATE** program. Select the required point in a dialogue box and save the stress distribution to a **".kon_spg"** file.

7.11 ? menu

7.11.1 "Copyright" menu item

You will see a copyright message and information on the program version number.

The **"System"** button shows information on your computer configuration and the folders used by **GGU-SLAB**.

7.11.2 "Maxima" menu item

A box opens showing the maximum values allowed by the program for the nodes and elements of the FEM mesh.

7.11.3 "Help" menu item

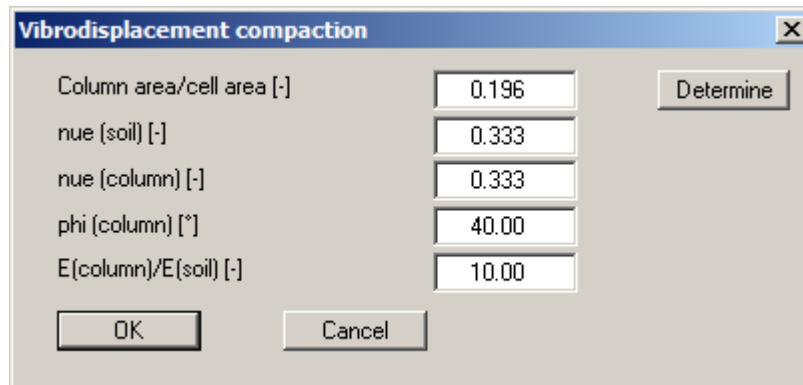
The **GGU-SLAB** manual is opened as a PDF document. The help function can also be accessed using the [**F1**] function key.

7.11.4 "Test slab reinforcement" menu item

If the appropriate input has been completed you can test the slab reinforcement using this menu item.

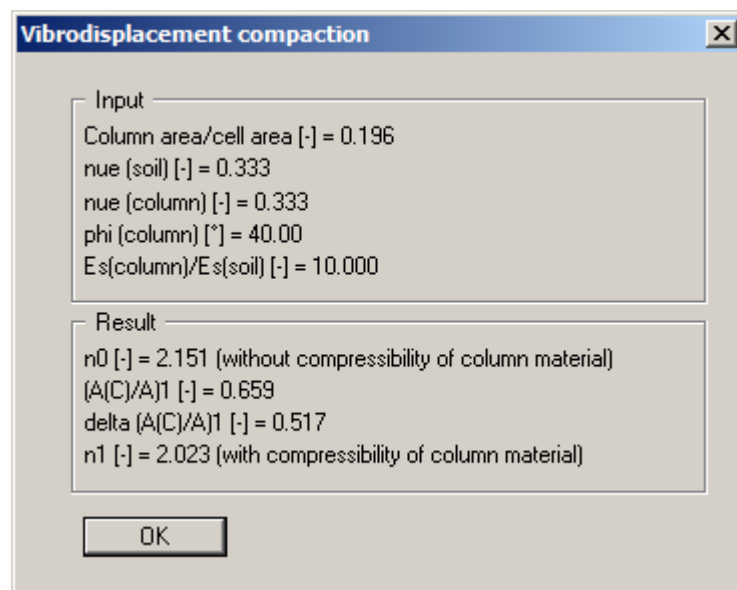
7.11.5 "Test vibrodisplacement compaction" menu item

The improvement brought about by vibrodisplacement compaction can be tested by varying the soil properties using this menu item.



Parameter	Value
Column area/cell area [-]	0.196
nue (soil) [-]	0.333
nue (column) [-]	0.333
phi (column) [°]	40.00
E(column)/E(soil) [-]	10.00

Using the "**Determine**" button the column/cell area ratio " $A(C)/A$ " can be determined using a variety of array procedures. After confirming your input values using "**OK**", the results are presented in a message box:



Section	Parameter	Value
Input	Column area/cell area [-]	0.196
	nue (soil) [-]	0.333
	nue (column) [-]	0.333
	phi (column) [°]	40.00
	Es(column)/Es(soil) [-]	10.000
Result	n0 [-]	2.151 (without compressibility of column material)
	(A(C)/A)1 [-]	0.659
	delta (A(C)/A)1 [-]	0.517
	n1 [-]	2.023 (with compressibility of column material)

Click "**Cancel**" in the above dialog box to end the menu item.

7.11.6 "Rotate all" menu item

Once entered, the system can be rotated through a user-defined angle. It is also possible to increase or decrease the scale or to displace the system in the x- or y-directions. The boundary conditions ϕ_{ix} and ϕ_{iy} as well as m_{xx} and m_{yy} are not altered.

7.11.7 "GGU on the web" menu item

Using this menu item you can access the GGU Software website: www.ggu-software.com. Keep in touch with new program versions and the regular *download* offers.

If you would like to be automatically notified about program innovations, please register for the Newsletter in our Knowledge Base. Go to the following website: <http://kbase.civilserve.com>.

7.11.8 "GGU support" menu item

This menu item takes to the GGU-Software [Support area](#) at www.ggu-software.com.

7.11.9 "What's new?" menu item

You will see information on program improvements in comparison to older versions.

7.11.10 "Language preferences" menu item

This menu item allows you to switch the menus and graphics from German to English or Spanish and vice versa. The program always starts with the language setting applicable when it was last ended.

7.11.11 "Benchmark" menu item (constrained modulus method only)

Average computation times are displayed.

8 Tips and tricks

8.1 Keyboard and mouse

You can scroll the screen with the keyboard using the cursor keys and the **[Page up]** and **[Page down]** keys. By clicking and pulling with the mouse, with **[Ctrl]** pressed, you activate the zoom function, i.e. the selected section will fill the screen. Use the mouse wheel to zoom in or out of the screen view or to pan.

In addition, scale and coordinates of the system graphics (drawing area within the plotting margins) can be altered directly using the mouse wheel. The following mouse wheel functions are available

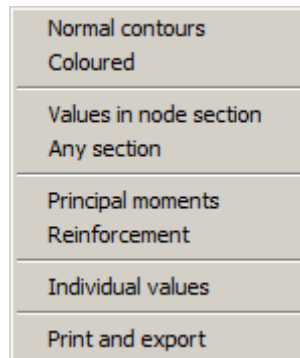
Change system graphics (new values can be checked in "Page size + margins/Manual resize (editor)"):

- **[Ctrl]** + mouse wheel up = enlarge system graphics (change of scale)
- **[Ctrl]** + mouse wheel down = shrink system graphics (change of scale)
- **[Shift]** + mouse wheel up = move system graphics up
(change in system coordinates)
- **[Shift]** + mouse wheel down = move system graphics down
(change in system coordinates)
- **[Shift]** + **[Ctrl]** + mouse wheel up = move system graphics right
(change in system coordinates)
- **[Shift]** + **[Ctrl]** + mouse wheel down = move system graphics left
(change in system coordinates)

Change screen coordinates:

- Mouse wheel up = move screen image up
- Mouse wheel down = move screen image down
- **[Alt]** + **[Ctrl]** + mouse wheel up = enlarge screen image (zoom in)
- **[Alt]** + **[Ctrl]** + mouse wheel down = shrink screen image (zoom out)
- **[Alt]** + **[Shift]** + mouse wheel up = move screen image right
- **[Alt]** + **[Shift]** + mouse wheel down = move screen image left

If you click the right mouse button anywhere on the screen a context menu containing the principal menu items opens.



By double-clicking the left mouse button on legends or **Mini-CAD** objects, the editor for the selected element immediately opens, allowing it to be edited.


You can simplify system input by integrating graphics or DXF files into the system via the **Mini-CAD** module (see "**Mini-CAD**" manual). The size of the graphics or the DXF data can be imported to the correct scale. However, this is not absolutely necessary for mesh generation. If you do not model the system to scale using a graphics file, you can perform a scale correction after input is complete using the menu item "**FEM mesh/Change (nodes)**", "**Via equation**" button.

8.2 Function keys

Some of the function keys are assigned program functions. The allocations are noted after the corresponding menu items. The individual function key allocations are:

- [Esc] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [F1] opens the manual file.
- [F2] refreshes the screen without altering the current magnification.
- [F3] opens the menu item "**FEM mesh/Define (FEM) nodes**".
- [F4] opens the menu item "**FEM mesh/Manual (FEM) mesh**".
- [F5] opens the menu item "**System/Analyse**".
- [F6] opens the menu item "**System/Reinforce slab**".
- [F7] opens the menu item "**FEM mesh/(Refine) All**".
- [F9] opens the menu item "**Page size + margins/Auto-resize**".
- [F11] activates the menu item "**Graphics preferences/Move legends**".

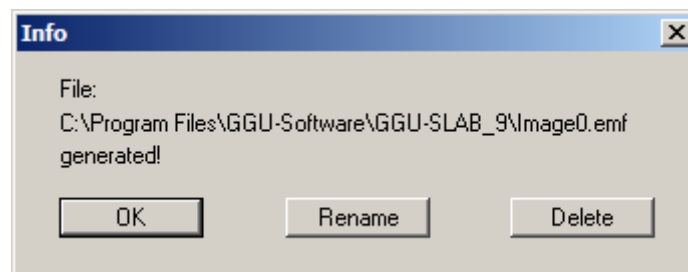
8.3 "Copy/print area" icon

A dialog box opens when the "**Copy/print area**" icon  in the menu toolbar is clicked, describing the options available for this function. For example, using this icon it is possible to either copy areas of the screen graphics and paste them into the report, or send them directly to a printer.

In the dialog box, first select where the copied area should be transferred to: "**Clipboard**", "**File**" or "**Printer**". The cursor is displayed as a cross after leaving the dialog box and, keeping the left mouse button pressed, the required area may be enclosed. If the marked area does not suit your requirements, abort the subsequent boxes and restart the function by clicking the icon again.

If "**Clipboard**" was selected, move to the MS Word document (for example) after marking the area and paste the copied graphics using "*Edit/Paste*".

If "**File**" was selected, the following dialog box opens once the area has been defined:



The default location of the file is the folder from which the program is started and, if several files are created, the file is given the file name "**Image0.emf**" with sequential numbering. If the "**Rename**" button in the dialog box is clicked, a file selector box opens and the copied area can be saved under a different name in a user-defined folder. Saving can be aborted by pressing the "**Delete**" button.

If the "**Printer**" button was pressed in the first dialog box, a dialog box for defining the printer settings opens after marking the area. Following this, a dialog box for defining the image output settings opens. After confirming the settings the defined area is output to the selected printer.

9 Worked slab analysis examples

9.1 Example 1: Analysis using the subgrade reaction modulus method

9.1.1 System description (Example 1)

Below, slab analysis is demonstrated using a practical example. The example is taken from

- H. R. Schwarz
Methode der finiten Elemente (*Finite-Element Methods*), Teubner, Stuttgart 1991

The example is shown in Section 6.1.5.2 on pages 391 pp of the book. It deals with a bridge slab with the following dimensions:

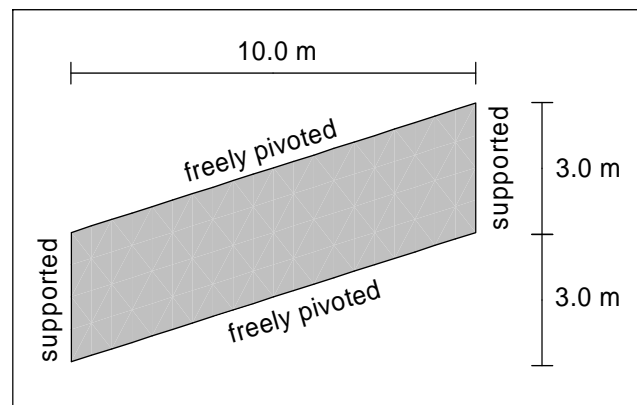


Figure 6 Example from Schwarz 1991

The following slab data are given:

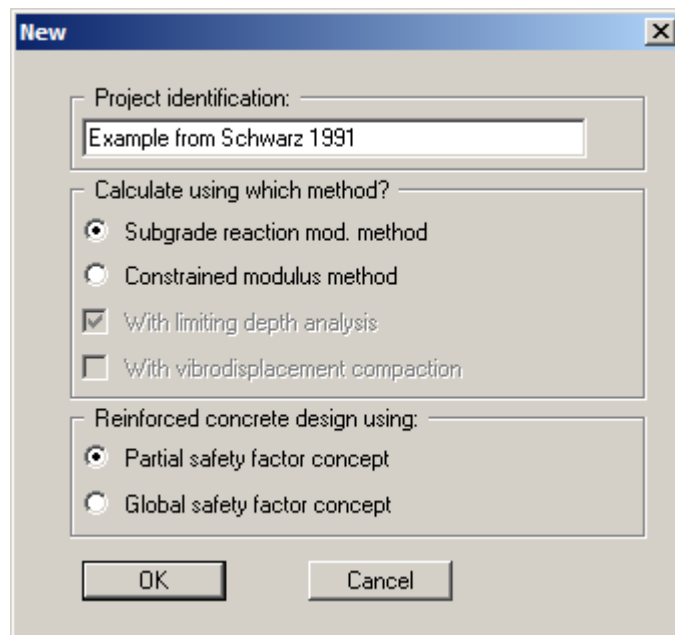
- thickness d of slab = 0.40 m
- Young's modulus of concrete = 40,000.0 MN/m²
- Poisson's ratio of concrete = $\frac{1}{6} = 0.1667$
- specific weight of concrete = 2.548 g/cm³ $\rightarrow \gamma = 25$ kN/m³

The loading on the slab results entirely from self weight.

Because this is a bridge slab the system is first analysed without subgrade.

9.1.2 Step 1: Select the analysis method (Example 1)

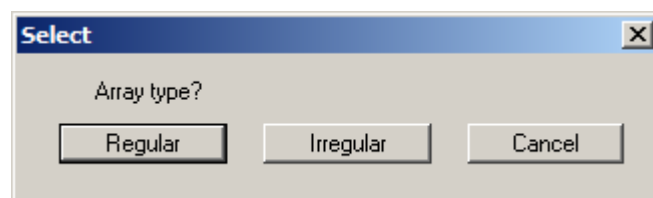
Start the program and go to the **"File/New"** menu item.



In the dialog box, select the **"Subgrade reaction mod. method"** button and **"Partial safety factor concept"**.

9.1.3 Step 2: Define the FEM mesh (Example 1)

A quadrilateral system is used. Mesh generation is simplest using the menu item **"FEM mesh/Array"**.



Press the **"Regular"** button first and then the **"Quadrilateral"** button.

Enter the data as given in the dialog box. Activate the **"Delete current nodes"** check box. The immovable left and right system boundaries can be defined in the above dialog box by activating the **"Right"** and **"Left"** check boxes. If all of the check boxes are deactivated (= all boundaries are assigned a free-earth support), define the immovable boundaries in Step 5 (see Section 9.1.6).

The slab is displayed. Go to the menu item **"Page size + margins/Auto-resize"** in order to achieve a screen-filling representation of the system.

9.1.4 Step 3: Optimise the FEM mesh diagonals (Example 1)

Normally, an optimal FEM mesh is generated. For this example, go to the menu item **"FEM mesh/Optimise"** and press the **"Diagonals"** button. The mesh then corresponds to the example used by Schwarz.

9.1.5 Step 4: Assign the subgrade reaction modulus (Example 1)

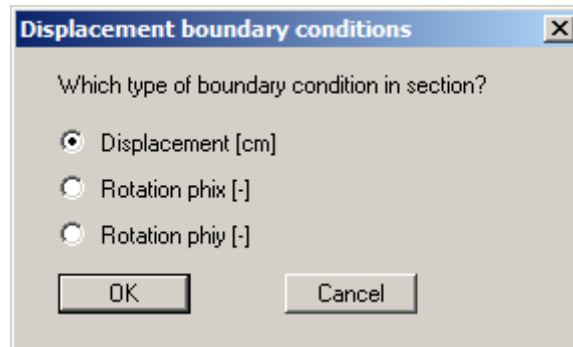
Go to the menu item **"Subgrade/Default subgrade"**.

Enter a **0** as subgrade and press the **"For all"** button. This assigns all nodes the same subgrade.

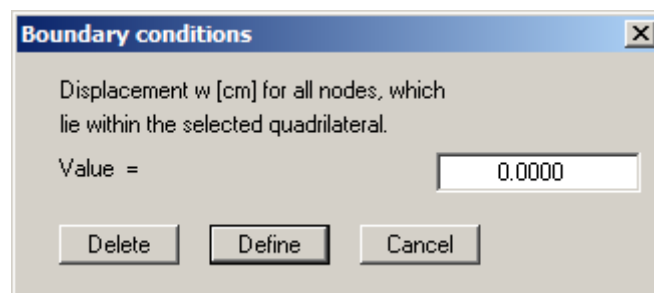
9.1.6 Step 5: Assign displacement boundary conditions (Example 1)

If you have already generated the immovable boundaries at the left and right of the system using the array function while defining the FEM mesh, you may skip this step.

Otherwise, go to the menu item "**Boundary/(Displacement) In section**".



Activate the "**Displacement [m]**" option button and leave the box using "**OK**". Then encompass the left 5 nodes of the system with the quadrilateral. The following dialog box then opens:



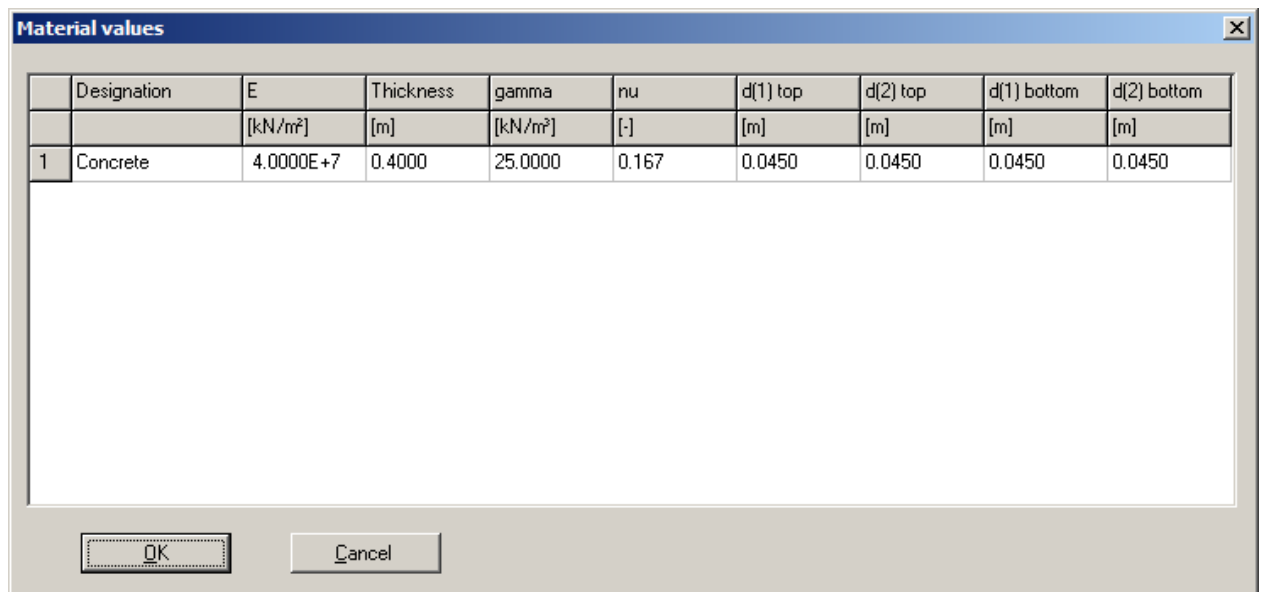
Press the "**Define**" button. This assigns a vertically immovable support to the left boundary. Do the same for the right 5 systems nodes. The free boundaries at the top and bottom slab faces do not need defining separately. The case of a free-earth support applies at all boundaries at which no action or displacement boundary condition is defined (also see Section 6.2).

If you define your FEM mesh using the "**FEM mesh/Array**" function you can specify immovable boundaries in the preferences for the node array by activating the corresponding option buttons for the sides required (see Section 9.1.3, "**Step 2: Define the FEM mesh**").

If you wish, you can check the entered boundary conditions by pointing to the menu item "**Boundary/Check**".

9.1.7 Step 6: Assign slab material (Example 1)

Go to the "System/Slab material" menu item.



The "Material values" dialog box contains a table with the following data:

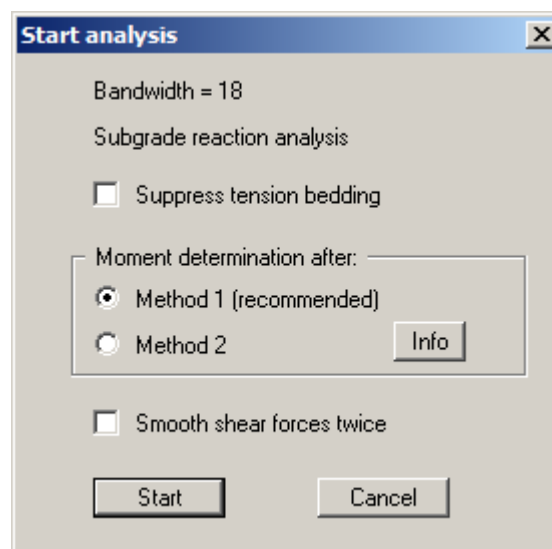
	Designation	E	Thickness	gamma	nu	d(1) top	d(2) top	d(1) bottom	d(2) bottom
		[kN/m²]	[m]	[kN/m²]	[-]	[m]	[m]	[m]	[m]
1	Concrete	4.0000E+7	0.4000	25.0000	0.167	0.0450	0.0450	0.0450	0.0450

At the bottom of the dialog box are "OK" and "Cancel" buttons.

Enter the values given in the dialog box. The numerical input for d(1)top, d(2)top, d(1)bottom and d(2)bottom are only important for subsequent slab reinforcement. This defines the distance of the centroid of the reinforcement from the upper and lower slab surface for the principal directions of reinforcement **1** and **2**.

9.1.8 Step 7: Analyse slab (Example 1)

Go to the menu item "System/Analyse" and if necessary optimise the bandwidth. Press the "Start" button.



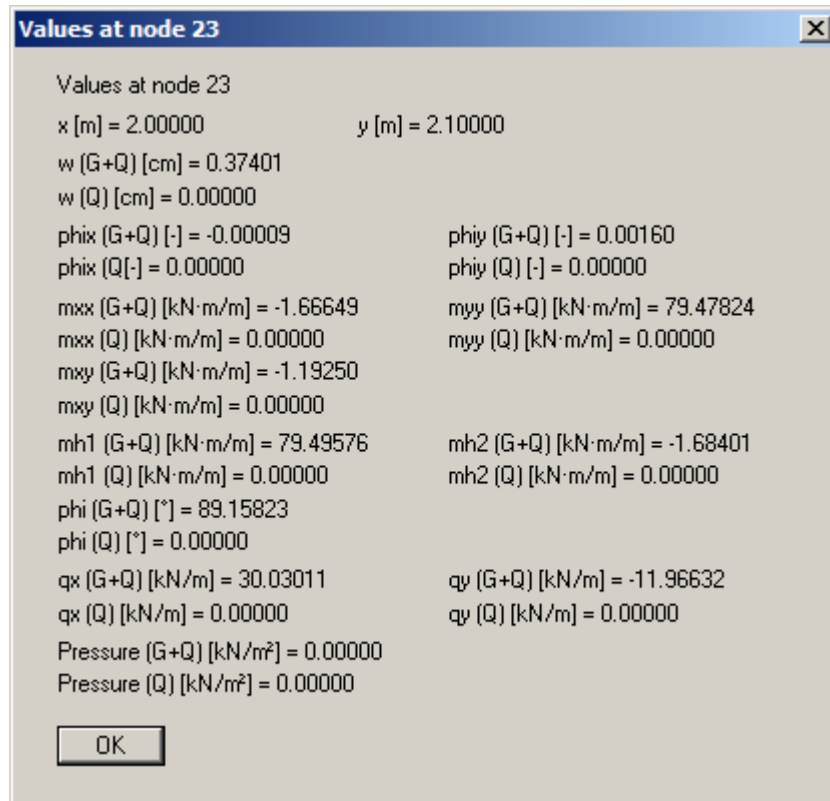
The "Start analysis" dialog box contains the following settings:

- Bandwidth = 18
- Subgrade reaction analysis
- ☐ Suppress tension bedding
- Moment determination after:
 - ☒ Method 1 (recommended)
 - ☐ Method 2
- ☐ Smooth shear forces twice

At the bottom are "Start" and "Cancel" buttons. An "Info" button is located next to the moment determination methods.

9.1.9 Step 8: Evaluation via individual values (Example 1)

In order to compare the results to those of Schwarz go to the menu item "**Evaluation/Individual values**". If you click the required node using the left mouse button you will see a result box for this nodes, e.g. for the node in the slab centre at $x = 2.0$ m:



For the slab centre and the lower slab face for the values of $x = 2, 4, 6$ and 8 m Schwarz gives the displacements listed in the following tables. The results ascertained using **GGU-SLAB** (converted to [m]) are listed beside them.

x [m]	y [m]	Schwarz's solution	GGU-SLAB result
2,0	2,1	0,00373	0,00374
4,0	2,7	0,00604	0,00606
6,0	3,3	0,00604	0,00606
8,0	3,9	0,00373	0,00374

Table 1 Displacements at the slab centre [m]

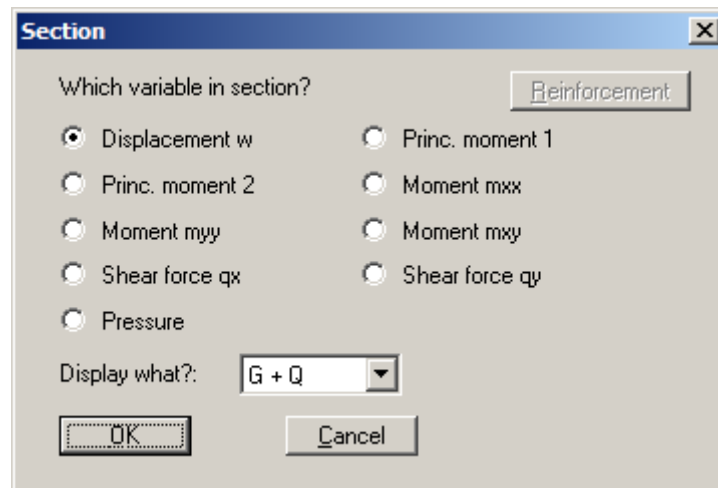
x [m]	y [m]	Schwarz's solution	GGU-SLAB result
2,0	0,6	0,00367	0,00368
4,0	1,2	0,00608	0,00608
6,0	1,8	0,00623	0,00624
8,0	2,4	0,00396	0,00396

Table 2 Displacements at the lower slab face [m]

The results are practically identical.

9.1.10 Step 9: Drawing the deflection curve and other evaluations (Example 1)

Finally, the deflection curve will be drawn through the slab centre. Go to the menu item "**Evaluation/Values in node section**" and click the central node on the left slab boundary and then the central node on the right slab boundary. To conclude, press [**Return**].



Activate the "**Displacement w**" option button and confirm using "**OK**". You can specify a number of preferences for the visualisation. The deflection curve is then displayed.

Go to the menu item "**File/Print and export**" if you want to print the deflection curve. You can display contours of displacements, moments, etc. in a similar manner (menu item "**Evaluation/Normal contours**", etc.) or determine the support forces (menu item "**Evaluation/Support**"). All node values can be annotated with the results in table form (menu item "**Evaluation/Table**"). You can generate circle charts displaying the respective state variables as circles of varying (value-dependent) size (menu item "**Evaluation/Circles**"). You can annotate the graphics using the "**Mini-CAD**" module or create a sheet header using the "**Header CAD**" module. Please see the Mini-CAD manual.

You can now reinforce the slab to DIN 1045 (menu item "**System/Reinforce slab**").

Save your analysis results to a file before carrying on (menu item "**File/Save as**").

9.2 Example 2: Slab with constant subgrade

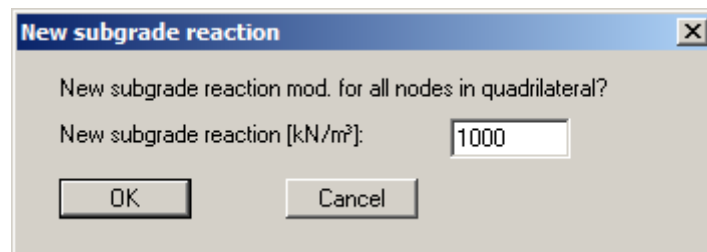
9.2.1 System description (Example 2)

The bridge slab in Example 1 will now be seriously *abused* by:

- applying a subgrade of 1000 kN/m³ at all nodes,
- eliminating support conditions at the left and right boundaries and
- applying a point load of 100 kN at the points (x = 3.0; y = 2.4) and (x = 7.0; y = 3.6) respectively.

9.2.2 Step 1: Change the subgrade reaction modulus (Example 2)

Go to the "Subgrade/In section" menu item and activate the "Display subgrade reaction" check box. Encompass the entire system with the quadrilateral, ensuring that all nodes are within.

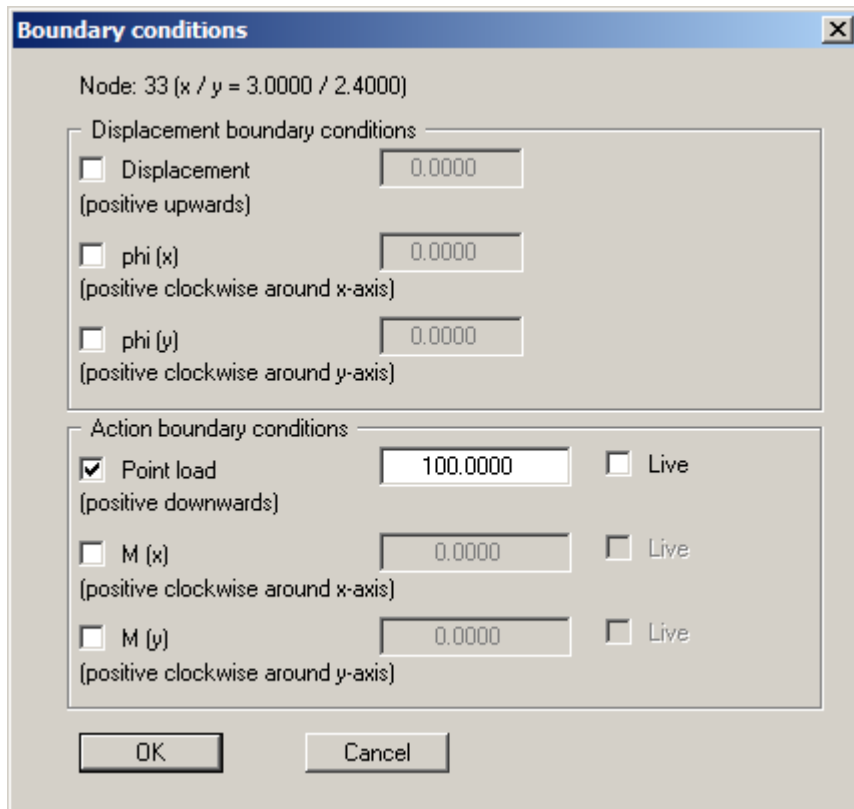


Enter **1000** for the subgrade reaction modulus and confirm with "**OK**". All nodes have now been assigned this subgrade reaction modulus. This could also have been done (and perhaps even easier) using the menu item "Subgrade/Default subgrade" and then the "For all" Button.

9.2.3 Step 2: Change the boundary conditions (Example 2)

Delete the boundary conditions at the left and right slab boundary. Go to the "**Boundary/(Displacement) In section**" menu item. Encompass the entire system with the quadrilateral and press the "**Delete**" button in the dialog box.

Now define the point loads. Go to the "**Boundary/Point loads**" menu item. Click the nodes with the two coordinates given above (node 33 and node 73) and enter the following:



The image shows a software dialog box titled "Boundary conditions" with a close button (X) in the top right corner. The dialog is for "Node: 33 (x / y = 3.0000 / 2.4000)". It is divided into two main sections: "Displacement boundary conditions" and "Action boundary conditions".

Displacement boundary conditions:

- ☐ Displacement (positive upwards) with a text input field containing "0.0000".
- ☐ phi (x) (positive clockwise around x-axis) with a text input field containing "0.0000".
- ☐ phi (y) (positive clockwise around y-axis) with a text input field containing "0.0000".

Action boundary conditions:

- ☒ Point load (positive downwards) with a text input field containing "100.0000" and a ☐ Live checkbox.
- ☐ M (x) (positive clockwise around x-axis) with a text input field containing "0.0000" and a ☐ Live checkbox.
- ☐ M (y) (positive clockwise around y-axis) with a text input field containing "0.0000" and a ☐ Live checkbox.

At the bottom of the dialog are two buttons: "OK" and "Cancel".

Now analyse the system and evaluate it, as previously described above.

9.3 Example 3: Analysis using the constrained modulus profile method

9.3.1 System description (Example 3)

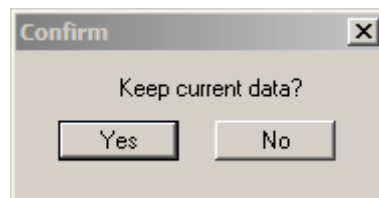
The system analysed in the previous section will now be analysed again using the constrained modulus method. The slab is supported on a surface with the following constrained modulus profile:

- to 3.0 m below slab: $E_s = 1,000 \text{ kN/m}^2$,
- to 7.0 m below slab: $E_s = 20,000 \text{ kN/m}^2$.

The largely non-deformable bedrock lies at 7.0 m below slab.

9.3.2 Step 1: Select the analysis method (Example 3)

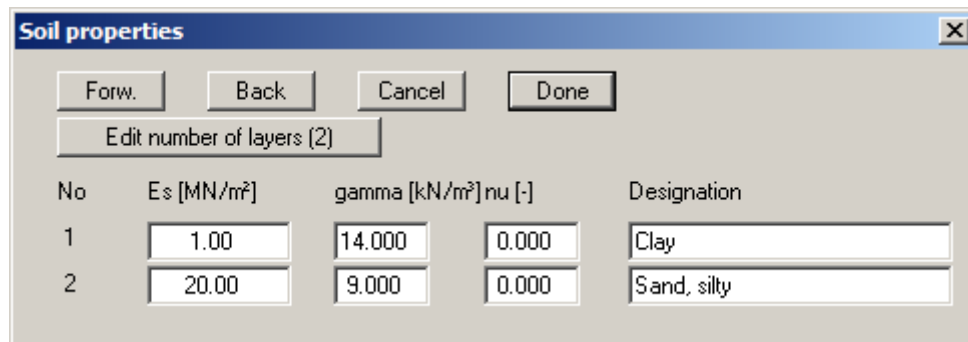
Go to the "File/New" menu item and activate the "Constrained modulus method" and "With limiting depth analysis" option buttons. Confirm the query



with "Yes" to get the mesh. The "Constrained mod." menu now appears in place of the "Sub-grade" menu.

9.3.3 Step 2: Define the constrained modulus profile (Example 3)

Select the menu item "**Constrained mod./Layers**". Press the "**Edit number of layers**" and enter a **2**, because we have 2 layers in the system.



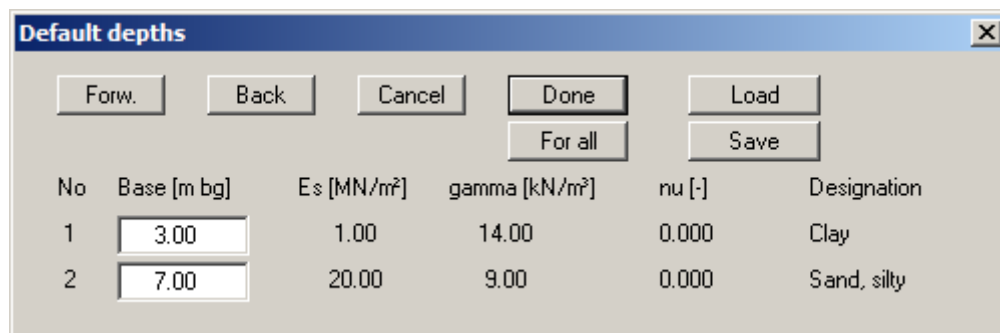
The "Soil properties" dialog box contains buttons for "Forw.", "Back", "Cancel", and "Done". Below these is a button labeled "Edit number of layers [2]". The main area is a table with four columns: "No", "Es [MN/m²]", "gamma [kN/m³]", and "nu [-]". The "Designation" column is a text field. Layer 1 has Es=1.00, gamma=14.000, nu=0.000, and Designation="Clay". Layer 2 has Es=20.00, gamma=9.000, nu=0.000, and Designation="Sand, silty".

No	Es [MN/m²]	gamma [kN/m³]	nu [-]	Designation
1	1.00	14.000	0.000	Clay
2	20.00	9.000	0.000	Sand, silty

Enter the values given in the dialog box. In this case "**nu**" represents the Poisson's ratio of the soil.

Input of the unit weight of the soil is necessary for the limiting depth analysis. If a water level is present within a single layer, the different unit weights can be taken into consideration by dividing the layer into two. Enter the weight of the wet soil for the area above the groundwater table and the buoyant weight of the soil below the groundwater table.

Select the "**Constrained mod./Default depths**" menu item and enter the values from the following dialog box:



The "Default depths" dialog box contains buttons for "Forw.", "Back", "Cancel", "Done", "Load", "For all", and "Save". The main area is a table with six columns: "No", "Base [m bg]", "Es [MN/m²]", "gamma [kN/m³]", "nu [-]", and "Designation". Layer 1 has Base=3.00, Es=1.00, gamma=14.00, nu=0.000, and Designation="Clay". Layer 2 has Base=7.00, Es=20.00, gamma=9.00, nu=0.000, and Designation="Sand, silty".

No	Base [m bg]	Es [MN/m²]	gamma [kN/m³]	nu [-]	Designation
1	3.00	1.00	14.00	0.000	Clay
2	7.00	20.00	9.00	0.000	Sand, silty

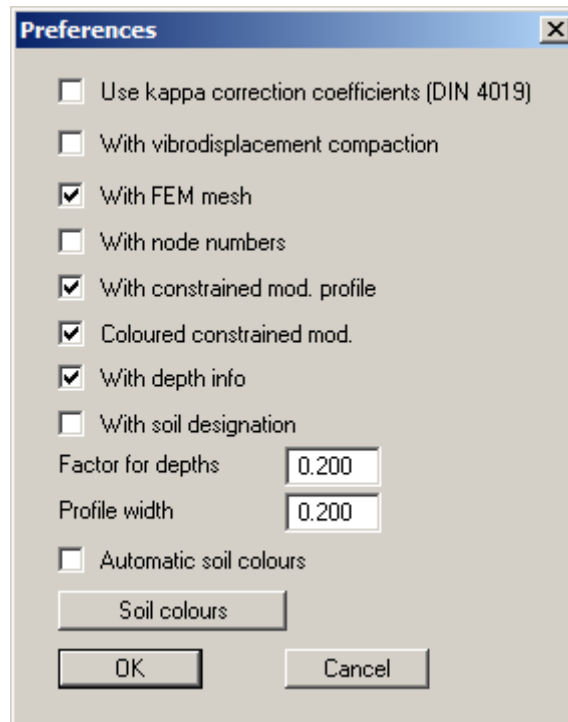
The depths entered here correspond to those specified at the beginning of this section.

9.3.4 Step 3: Define the constrained modulus profile mesh (Example 3)

Now go to the menu item "**Constrained mod./Define nodes**" (not in the "**FEM mesh**" menu!) and click four points, located roughly at:

- constrained modulus profile 1 at $x = -1$ and $y = -1$,
- constrained modulus profile 2 at $x = -1$ and $y = +4$,
- constrained modulus profile 3 at $x = +11$ and $y = 2$,
- constrained modulus profile 4 at $x = +11$ and $y = 7$

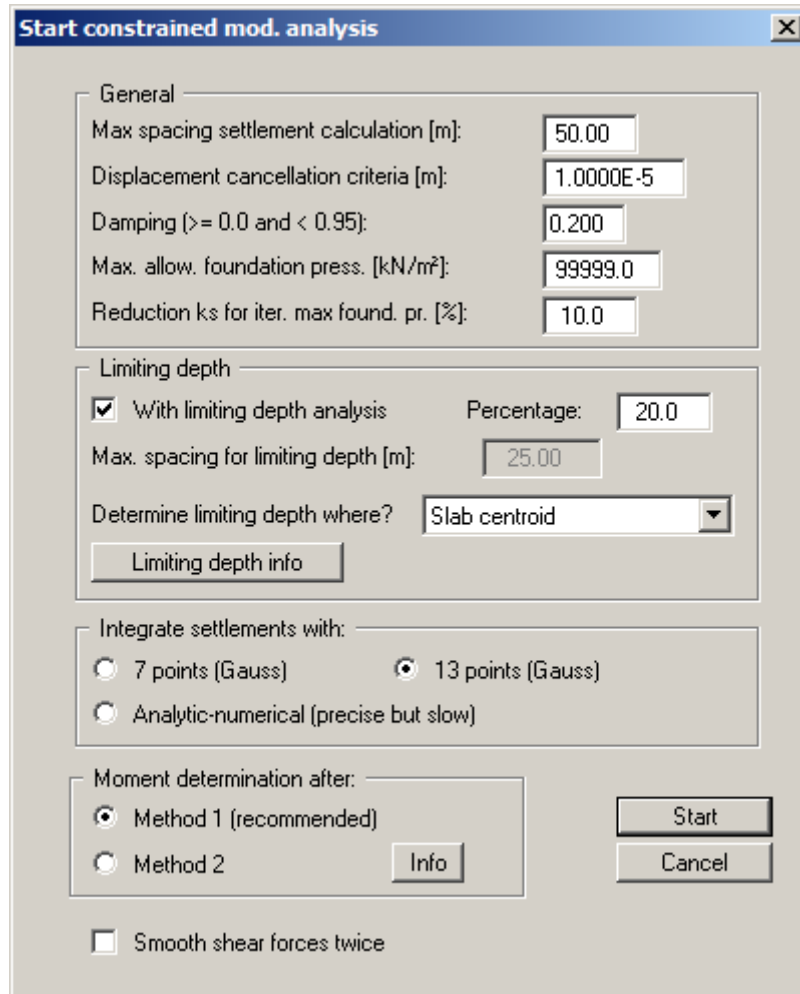
These coordinates ensure that all of the slab's nodes are included. Using the menu item "**Page size + margins/Auto-resize**" or, more simply, by pressing [**F9**], a screen-filling representation can be achieved if the point in question lies outside of the current view. The drilling points (constrained modulus profile) are displayed parallel to input with the mouse. The profiles look a bit rough in comparison to the slab. If necessary, correct the optics by going to the menu item "**Constrained mod./Preferences**" and making the following input:



Now connect the four constrained modulus profiles to form a mesh. To do this, go to the menu item "**Constrained mod./Automesh**" (not the "**FEM mesh**" menu!).

9.3.5 Step 4: Analyse slab (Example 3)

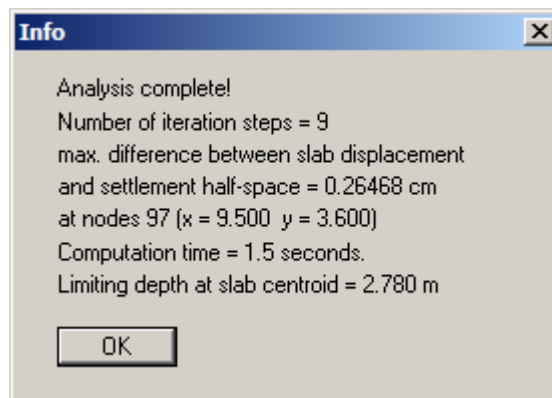
All necessary input is now complete and you can start the analysis. It may be necessary to optimise the bandwidth first.



The dialog box titled "Start constrained mod. analysis" contains the following sections and controls:

- General**
 - Max spacing settlement calculation [m]: 50.00
 - Displacement cancellation criteria [m]: 1.0000E-5
 - Damping (≥ 0.0 and < 0.95): 0.200
 - Max. allow. foundation press. [kN/m²]: 99999.0
 - Reduction ks for iter. max found. pr. [%]: 10.0
- Limiting depth**
 - ☒ With limiting depth analysis Percentage: 20.0
 - Max. spacing for limiting depth [m]: 25.00
 - Determine limiting depth where? Slab centroid (dropdown menu)
 - Limiting depth info (button)
- Integrate settlements with:**
 - ☐ 7 points (Gauss)
 - ☒ 13 points (Gauss)
 - ☐ Analytic-numerical (precise but slow)
- Moment determination after:**
 - ☒ Method 1 (recommended)
 - ☐ Method 2 Info (button)
- ☐ Smooth shear forces twice
- Start (button)
- Cancel (button)

You do not need to alter anything in the start dialog box. A detailed description can be found in the corresponding menu items (see Section 7.6.6.3). When analysis is complete you will initially see the message box shown below. After this final message you can evaluate the system as described in the examples and the explanations for the items of the **"Evaluation"** and **"Special"** menus.



The "Info" message box displays the following text:

Analysis complete!
Number of iteration steps = 9
max. difference between slab displacement
and settlement half-space = 0.26468 cm
at nodes 97 (x = 9.500 y = 3.600)
Computation time = 1.5 seconds.
Limiting depth at slab centroid = 2.780 m

OK (button)

10 Index

A

Action boundary condition, define/delete	
for individual FEM nodes	55
Analysis method, select	20
Analysis results, save in file	21
Analysis time, limit for settlement analysis	63
Angle of rotation, define for 3D graphics	70
Approximation method	16
Array, irregular	34
Array, regular	33
Axial stiffness of pile, consider	59

B

Bandwidth	62
Beams material, assign numbers	58
Beams material, define parameters	61
Beams, assign material numbers	58
Beams, define	58
Beams, define material parameters	61
Beams, delete all	58
Beams, delete individual	58
Beams, display	91
Beams, display in legend	73
Boundary conditions, free-earth support	18, 55
Boundary conditions, natural	18
Boussinesq equation	18

C

Centroid of the reinforcement	60
Cholesky	62
Circle chart, representation preferences	83
Clipboard	27
CodeMeter stick	9
Colour bar, contours	49
Colour, define for constrained mod. profiles	46
Colour, define for slab material	71
Colour, define for soils of constrained	
mod. profile	72
Colours, interpolation/FEM mesh	42
Colours, switch on/off	69
Colours/pens, define for graphical elements	68
Company letterhead, add via Mini-CAD	68
Constrained mod. profile mesh, define	
nodes using editor	52
Constrained mod. profile mesh, define/	
delete elements using mouse	53
Constrained mod. profile mesh, define/	
delete nodes using mouse	51
Constrained mod. profile mesh, delete	53
Constrained mod. profile mesh, edit	
individual node	52
Constrained mod. profile mesh, generate	
automatically	53
Constrained mod. profile mesh, generate	
automatically with default depths	51
Constrained mod. profile mesh, move	
individual node using mouse	52

Constrained mod. profile mesh, refine	
elements	53
Constrained mod. profile, display in legend	72
Constrained mod. profile, edit individual	
layer depths	49
Constrained mod. profiles, assign default	
layer depths	47
Constrained mod. profiles, contours of	
layer bases	48
Constrained mod. profiles, define colours	46
Constrained mod. profiles, define layer	
depths in advance	47
Constrained mod. profiles, define layers	46
Constrained mod. profiles, define	
soil properties	46
Constrained mod. profiles, display	
individual profile	49
Constrained mod. profiles, duplicate layer	50
Constrained mod. profiles, edit/assign	
layer depths retroactively	50
Constrained mod. profiles, presentation	
parameters	45
Context menu, open	102
Contour lines, dashing	80
Contours, differences between settlements	
and displacements	94
Contours, layer bases of constrained	
mod. profiles	48
Contours, preferences for 3D array	
presentation	82
Contours, preferences for 3D presentation	80
Contours, preferences for coloured-filled	
lines	48
Contours, preferences for normal lines	79
Contours, select state variable	78
Contours, subgrade reaction moduli of	
FEM mesh	42
Contours, subgrade reaction moduli of	
interpolation mesh	44
Contours, subgrade reaction moduli	
resulting from iteration	94
Convergence of solution	64
Coordinates FEM mesh, import as	
ASCII file	21
Coordinates FEM mesh, save to ASCII file	22
Coordinates of FEM mesh, import via	
Windows clipboard	31
Coordinates, alter via editor	75
Coordinates, alter with mouse	75
Coordinates, edit for individual FEM nodes	41
Coordinates, optimise/reset	75
Coordinates, save/load	75
Copy/print area	27, 69, 103
Cutting borders, switch on/off	77

D

Damping	64
Dataset description, enter	20
Default subgrade, enter	41
Deflection curve, display	110

Delauney triangulation	36
Delete, all FEM nodes	31
Delete, beams	58
Delete, constrained mod. profile mesh	53
Delete, distributed loads	57
Delete, individual elements of constrained mod. profile mesh	53
Delete, individual FEM elements	36
Delete, line loads	56
Delete, several FEM elements	37
Delete, single springs	59
Differences between settlements and displacements, display as contours	94
DIN 1045/DIN 1045-1	65
DIN 4019	64
Displacement assumption to Zienkiewicz	17
Displacement boundary condition, define/ delete for individual FEM nodes	55
Displacement boundary condition, define/ delete for several FEM nodes	56
Displacement, sign rule	19
Distributed load, determine from $M + V$ for all FEM elements	36
Distributed loads, define for certain FEM elements	57
Distributed loads, define for certain material ..	57
Distributed loads, raise/reduce	57
Drawing area, define	76
DXF file, export	27
DXF file, import	8

E

Editor window, output table	25
Elastic-isotropic half-space	63
EMF format	27
Equation solver	62
Evaluation, select state variable for contour lines	78
Export, stress distributions to GGU-CONSOLIDATE	97

F

FEM elements, refinement methods	39
FEM mesh, define FEM elements using mouse	36
FEM mesh, delete FEM elements automatically	37
FEM mesh, delete FEM elements using mouse	36
FEM mesh, display contours of subgrade reaction moduli	42
FEM mesh, edit presentation preferences	29
FEM mesh, generate automatically	36
FEM mesh, optimise FEM elements	37
FEM mesh, refine a number of FEM elements	38
FEM mesh, refine certain FEM elements	39
FEM mesh, refine individual FEM elements ..	38
FEM mesh, show maximum values	98
FEM nodes, adjust scale	31
FEM nodes, assign default subgrade modulus	41

FEM nodes, define all using editor	31
FEM nodes, define for circular slabs	34
FEM nodes, define subgrade modulus in advance	41
FEM nodes, define using array	32
FEM nodes, define using mouse	30
FEM nodes, edit coordinates individually ..	32, 41
FEM nodes, edit subgrade reaction modulus ..	41
FEM nodes, import coordinates via Windows clipboard	31
FEM nodes, move all using editor	31
FEM nodes, move individually using mouse ..	32
FEM nodes, move selected using editor	32
Fence diagram, display state variables at FEM node	85
Fence diagram, display state variables between any points	86
File, display name in legend	70
File, load/save	20
Finite-difference method	16
Finite-element method	16
Font size, define for beam legend	73
Font size, define for constrained mod. profile legend	72
Font size, define for general legend	70
Font size, define for reinforcement legend	73
Font size, define for section legend	73
Font size, define for slab material legend	71
Font size, edit for drawing elements	76
Footer, output table	23
Footing bases, define for slab materials	66
Footing beams, consider	58
Free-earth support	18, 55
Function keys	102

G

Gamma, define above/below groundwater	114
Gamma, define for limiting depth analysis	46
Gaussian numerical integration	64
General page informations, add via Mini-CAD	68
Geological section, display	50
GGUCAD file, export	27
GGU-CONSOLIDATE file, export	97
GGUMiniCAD file, export	28
GGU-SETTLE	53
Graphics, add via Mini-CAD	68

H

Header CAD, use	68
Header, output table	23
Hooke's Law	16

I

Individual values, display	93
Installation	9
Interpolation mesh, contours of subgrade reaction moduli	44
Interpolation mesh, define for subgrade reaction mod. distribution	42
Interpolation mesh, edit mesh	43

Interpolation mesh, edit nodes	43
Interpolation mesh, modify subgrade reaction moduli using constants	43
Iteration process, constrained modulus method.....	18, 63
Iteration, cancellation criteria	63

K

Kappa correction coefficients, consider in settlement analysis	45
Knowledge Base, access	100

L

Language preferences	9, 100
Layout, define for output sheet	76
Layout, define for output table	23
Legends, move with mouse	73
Licence protection	9
Limiting depth analysis, define unit weight of soil	46
Limiting depth, activate analysis	20, 64
Limiting depth, display as contours.....	94
Line loads, define	56
Live load, define point load as.....	55
Live load, distributed load as.....	57
Live load, line load as.....	56

M

Manual, open as PDF file	98
Material colour/number, define for slab	71
Max. foundation pressure, define for analysis.....	64
Maximum values, show for FEM mesh	98
Metafile, export	27
Mini-CAD file, export	28
Mini-CAD, display objects behind system graphics	77
Mini-CAD, use	68
Minimum reinforcement, define.....	79
Moment, determination methods	17
Moment, sign rule.....	19
Mouse click functions.....	102
Mouse wheel functions.....	101

N

Navigation, output table	69
nu, deactivate display in legend.....	72
nu, define	46

O

Output table navigation	24
Output table, edit output as ASCII	25
Output table, edit output as graphics	23
Output table, select output format	22
Output table, switch to system graphics	24, 69
Overall spring constant C, calculation.....	59

P

Page format, define.....	76
Page margins, define.....	76
Page margins, switch on/off	77
Page section, copy/print.....	69, 103
Pagination, automatic	24, 25
Pen preferences.....	68
Pile foundation, consider via single springs	59
Plot margins, define.....	76
Point loads, define	55
Point loads, sign rule	19
Poisson's ratio, deactivate display in legend.....	72
Poisson's ratio, define	46
Principal moments, display.....	89
Print, graphics.....	26
Print, output table	27
Print, section	27, 69, 103
Print, several files	28
Printer preferences.....	25, 26
Program, display name in legend.....	70
Program, save/load preferences	74
Program, show improvements	100
Program, show information	98
Project data, add via Mini-CAD	68
Project identification, display	70
Project identification, enter.....	20, 60

Q

Quality of the approximation.....	18
Quality of the solution	16

R

Radius ratio, FEM elements	37
Refinement methods	39
Reinforcement directions, define.....	65
Reinforcement slab, select safety factor concept	65
Reinforcement, display.....	90
Reinforcement, select parameters for circle visualisation	83
Reinforcement, select parameters for table visualisation	84
Reinforcement, select variable for contour lines.....	78
Reinforcement, show in legend	73
Reinforcement, test.....	98
Resistance-settlement diagram to DIN 4014	59
Rotation, sign rule	19

S

Safety factor concept, select for reinforcement	20, 65
Scale, alter with mouse.....	75
Scale, define via editor	75
Scale, determine automatically	75
Scale, edit via factor	100
Scroll the screen	101
Section course, display in legend.....	73
Section values, calculate for settlement depression.....	97

Section, display state variables at FEM nodes	85	Subgrade reaction moduli, edit for individual FEM nodes	41
Section, display state variables between any points	86	Subgrade reaction moduli, edit for several FEM nodes	42
Section, display stresses between any points	95	Sum V, display results in a message box	93
Section, fence diagram preferences	86	Support forces, display as circle chart	88
Sectional values, calculate for fence diagram	86	System coordinates, alter via editor	75
Sectional values, calculate for stress distribution	95	System coordinates, alter with mouse	75
Settlement analysis, consider kappa correction coefficients	45	System coordinates, optimise/reset	75
Settlement analysis, limit analysis time	63	System coordinates, save/load	75
Settlement analysis, via data export to GGU-SETTLE	53	System input, using DXF/BMP files	30
Settlement depression, display	97	System properties, display in legend	70
Settlement, display for any points	93	System, analyse	62
Shear force, calculation	15, 17	System, rotate/displace	100
Shear reinforcement, display	90	System, show information	98
Sign	19	System, test	61
Single spring size C_B , calculation	59		
Single springs, define/delete	59	T	
Single springs, display	92	Tables, settings for visualisation	84
Slab material define parameters	60	Tension bedding, suppress	62
Slab material, assign numbers for individual FEM elements	58	Toolbar, edit for 3D graphics	70
Slab material, assign numbers for several FEM elements	58	Toolbar, edit for menu items	69
Slab material, display outline	29	Torsional stiffness, adopt moments	36
Slab material, display properties in legend	71	Torsional stiffness, start analysis	87
Slab, assign material numbers for individual FEM elements	58	Translation, activate	100
Slab, assign material numbers for several FEM elements	58	True-type font	68
Slab, define material parameters	60		
Slab, display material properties in legend	71	U	
Smart icons, for 3D graphics	70	Unit weight of soil, define above/below groundwater	114
Smart icons, for menu items	69	Unit weight of soil, define for limiting depth analysis	46
Smoothing out procedures, contours	79	Units, define for constrained mod./ displacement	60
Soil, colour/number, define for constrained mod. profile	72		
Spring constant of pile C_p , calculation	59	V	
State variables, display results of one FEM node	93	Version number, display in a message box	98
Status bar main program, activate	69	Version number, display in legend	70
Stress distribution at depth	96	Vertical settlements, display as contours in section	97
Stress distribution, export	97	Vibrodisplacement compaction, activate adoption	20, 45
Stress peaks	64	Vibrodisplacement compaction, activate explanation in legend	72
Stresses, display as contours in section	96	Vibrodisplacement compaction, enter soil properties	47
Stresses, display in section	95		
Subgrade reaction mod. distribution, display results of iteration	94	W	
Subgrade reaction mod. distribution, representation for FEM mesh	42	What you see is what you get	67
Subgrade reaction mod. distribution, representation for interpolation mesh	44		
Subgrade reaction moduli, assign from interpolation mesh	42, 44	Z	
Subgrade reaction moduli, edit for all FEM nodes	41	Zienkiewicz	8, 17
		Zoom factor, define for full-screen display	67
		Zoom function, activate	67, 68, 69, 101